

FieldView User's Guide

**SOFTWARE RELEASE
VERSION 2023**

11/7/2023

Tecplot, Inc.

Copyright ©2023 Tecplot, Inc.
Revision 2023
All Rights Reserved
Printed in USA
First Printing November 2001

FieldView is a registered trademark of Tecplot, Inc.

Tecplot, Inc. reserves the right to make changes in specifications and other information contained in this publication without prior notice and the reader should in all cases consult Tecplot, Inc. to determine whether any such changes have been made.

The terms and conditions governing the licensing of Tecplot, Inc. software consist solely of those set forth in the Tecplot, Inc. Binary License Terms and Conditions set forth by Tecplot, Inc.. No representations or other affirmation of fact contained in this publication, including but not limited to statements regarding capacity, response-time performance, suitability for use or performance of products described herein shall be deemed to be a warranty of Tecplot, Inc. for any purpose, or give rise to any liability by Tecplot, Inc. whatsoever.

In no event shall Tecplot, Inc. be liable for any incidental, indirect, special or consequential damages whatsoever (including but not limited to lost profits) arising out of or relating to this publication or the information contained in it, even if Tecplot, Inc. has been advised, knew or should have known of the possibility of such damages.

The software programs described in this document are confidential information and proprietary products of Tecplot, Inc. or its licensors.

Reprise License Manager (RLM) v12.4, Copyright (C) 2006-2018, Reprise Software, Inc.
Copyright (C) 2006-2018, Reprise Software, Inc. All rights reserved.
RLM Build 12.4.2

Reprise License Manager is a registered trademark or trademark of Reprise Software, Inc. in the U.S. and/or other countries.
Copyright (c) 2000 - 2021, Lawrence Livermore National Security, LLC
All rights reserved.

Copyright © 2002 GraphicsMagick Group, an organization dedicated to making software imaging solutions freely available.

Copyright © 1999 E. I. du Pont de Nemours and Company

Copyright © 2000-2002, Ghostgum Software Pty Ltd. All rights reserved.

Copyright © 2000 Markus Friedl. All rights reserved.

Copyright © 1999 - 2003 Bob Friesenhahn <bfriesen@simple.dallas.tx.us>

FIG: Facility for Interactive Generation of figures Copyright © 1985-1988 by Supoj Sutanthavibul Parts Copyright © 1989-2000 by Brian V.

Smith Parts Copyright © 1991 by Paul King

The Graphics Interchange Format © is the Copyright property of CompuServe Incorporated. GIF(sm) is a Service Mark property of CompuServe Incorporated.

Copyright © 1994 - 2000 by the Massachusetts Institute of Technology. All rights reserved.

Copyright © 1994-2000 TeCGraf, PUC-Rio. All rights reserved.

Copyright © 1991-2000 by Bell Labs Innovations for Lucent Technologies.

Copyright © 1998-2008 The OpenSSL Project. All rights reserved.

Copyright © 1995-1998 Eric Young (eay@cryptsoft.com) All rights reserved.

Copyright © 2002 Niels Provos <provos@citi.umich.edu> All rights reserved.

This product uses parts of the SFL package, Copyright © 1996-2000 iMatix Corporation 1991-2000 iMatix Corporation <<http://www.imatix.com>>.

Copyright © 1995-2002 Jean-loup Gailly and Mark Adler

Copyright © 1993 University of Chicago

Copyright © 1993 Mississippi State University

Copyright © 1999 Serika Kurusugawa. All rights reserved.

Copyright © 1999-2000 Mizi Research Inc. All rights reserved.

Copyright © 2001, 2002 Turbolinux, Inc. Written by James Su.

Copyright © 2000 Turbolinux, Inc. Written by Justin Yu and Sean Chen.

Copyright © 2000 Ming-Che Chuang

Copyright © 2002 WU Yi, HancornLinux Inc.

Copyright © 2000 Ming-Che Chuang

Copyright © 2001, 2002 ThizLinux Laboratory Ltd. Written by Anthony Fok.

Copyright © 2003-2004 immodule for Qt Project. All rights reserved.

Copyright © 2003-2006 Ben van Klinken and the CLucene Team

Copyright © 2008 Nokia Corporation and/or its subsidiary(-ies)

Copyright © 2004, 2005 Daniel M. Duley

Copyright © 2005 Bjoern Bergstroem

Copyright © 2005 Roberto Raggi

Copyright © The Internet Society (2001). All Rights Reserved.

Copyright © 1991 by AT&T.

Copyright © 2000 Hans Petter Bieker. All rights reserved.

Copyright © 1996 Daniel Dardailler.
 Copyright © 2002 USC/Information Sciences Institute
 Copyright © 2005-2007 Matthias Kretz <kretz@kde.org>
 Copyright © 1998 by Bjorn Reese <brees@imada.ou.dk>
 Copyright © 2000-2007 Gerard Juyn (gerard@libmng.com)
 Copyright © 2004, 2006-2008 Glenn Randers-Pehrson
 Copyright © 1996, 1997 Andreas Dilger
 Copyright © 1995, 1996 Guy Eric Schalnat, Group 42, Inc.
 Copyright © 2002 Robert Osfield.
 Copyright © 1998 Julian Smart, Robert Roebling [, ...]
 Copyright © 1999-2007 Brian Paul All Rights Reserved.
 Copyright © 2012 The FreeBSD Foundation. All rights reserved.
 Copyright © 2003-2018 University of Illinois at Urbana-Champaign. All rights reserved.
 Copyright 1992, 1993, 1994 Henry Spencer. All rights reserved.
 Copyright © 1989, 1991, 1993, 1994 The Regents of the University of California. All rights reserved.
 Copyright © 2009-2012, 2016 Daniel Stone
 Copyright © 2012 Ran Benita <ran234@gmail.com>
 Copyright © 2010, 2012 Intel Corporation
 Copyright © 2008, 2009 Dan Nicholson
 Copyright © 2010 Francisco Jerez <currojerez@riseup.net>
 Copyright © 1991-2000 Silicon Graphics, Inc. All Rights Reserved.
 Copyright 1987, 1988 by Digital Equipment Corporation, Maynard, Massachusetts.
 Copyright © 2011 Joseph Adams <joejadams3.14159@gmail.com>
 Copyright 1996 by Joseph Moss
 Copyright © Dmitry Golubev <lastguru@mail.ru>, 2003-2004
 Copyright © 2004, Gregory Mokhin <mokhin@bog.msu.ru>
 Copyright © 2006 Erdal Ronahi
 Copyright © 1998-2007 Free Software Foundation, Inc.
 Copyright © 2005, 2006, 2010 Free Software Foundation, Inc.
 Copyright © 1991-2010, Thomas G. Lane, Guido Vollbeding.
 This software is based in part on the work of the Independent JPEG Group.
 Copyright © 1988-1997 Sam Leffler
 Copyright © 1985, 1986, 1987, 1992 X Consortium
 Copyright 1996-2002 by David Turner, Robert Wilhelm, and Werner Lemberg
 Copyright © 2003 The XFree86 Project, Inc. All Rights Reserved
 Copyright © 1992 by Oki Technosystems Laboratory, Inc.
 Copyright © 1992 by Fuji Xerox Co., Ltd
 Copyright © 1995-2009 International Business Machines Corporation and others
 Copyright © 2001-2006 Bart Massey, Jamey Sharp, and Josh Triplett.
 Copyright © 1985, 1987-1988, 1990, 1993, 1994, 1998 The Open Group

This distribution contains PDF3D software copyright Visual Technology Services Ltd., all rights reserved.

This software is copyrighted by the Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, and other parties.

Development tools and related technology provided under license from

FieldView includes Open MPI, from the Open MPI Project (See README_Copyrights.txt)

Copyright © 2011-2017, Christopher C. Hulbert. All rights reserved.

FieldView includes the font Noto Sans CJK JP from the Google Noto Fonts (See README_Copyrights.txt)

Copyright © 2003-2017, Troy D. Hanson <http://troydhanson.github.com/uthash/>. All rights reserved.

VTK Reader: Copyright © 1993-2015 Ken Martin, Will Schroeder, Bill Lorensen. All rights reserved.

MAT-File Format © COPYRIGHT 1999-2017 by The MathWorks, Inc.

OSF, OSF/Motif[®], and Motif[®] are registered trademarks of The Open Software Foundation, Inc.

OpenGL is a trademark of Silicon Graphics, Inc. in the United States and other countries.

IBM is a registered trademark of International Business Machines Corporation.

Silicon Graphics is a registered trademark of Silicon Graphics, Inc.

Sun Microsystems is a registered trademark of Sun Microsystems, Inc.

UNIX is a registered trademark of AT&T.

PostScript is a registered trademark of Adobe Systems, Inc.

FIDAP, FLUENT and RAMPANT are registered trademarks of Ansys, Inc.

Phoenix is a registered trademark of Cham, Limited.

STAR-CD is a trademark of Computational Dynamics Ltd.

FLOW-3D[®] is a registered trademark of Flow Science, Inc.

CFX and CFX-TASCflow are registered trademarks of ANSYS-CFX.

MATLAB is a registered trademark of The MathWorks, Inc.

Noto is a trademark of Google Inc.

U.S. GOVERNMENT RESTRICTED RIGHTS. The Licensed Software is deemed to be commercial computer software as defined in FAR 12.212 and subject to restricted rights as defined in FAR Section 52.227-19 "Commercial Computer Software License", as applicable, and any successor regulations. Any use, modification, reproduction release, performance, display or disclosure of the Licensed Software by the U.S. Government shall be solely in accordance with the terms of the Tecplot, Inc. Binary License Terms and Conditions.

THIS DOCUMENT AND ALL INFORMATION CONTAINED HEREIN, ALONG WITH ALL FILE FORMATS, EXAMPLE DATA AND ONLINE DOCUMENTATION, IS INTENDED FOR THE EXCLUSIVE USE OF END-USER LICENSORS OF FieldView.

List of Figures 1**Chapter 1**

Starting FieldView.....	1
Running FieldView	1
Running Two Versions of FieldView Simultaneously	2
Remote FieldView Operation	2
Understanding the Console Window	7
Command Line Switches	9
Batch Mode	10
Hardware Render for Batch	10
Using Batch-only licensing	11
Running Restarts from the command line	12
Running Scripts from the command line	12
Running FVX from the command line	12
Running Python from the command line	13
DataGuideTM	13
PLOT3D/Fast Viewing Controls	16
Modifying Constants for PLOT3D Functions	16
Graphics Layout Size	17
FieldView Environment Variables	18
Possible Problems	24
Balance Between Memory Usage and Performance	25
Running Client-Server Mode	28
Server startup and control	30
Port Use	32
Data Reading via Client-server	33
Manual Server Startup	34
FieldView Parallelization with Multithreading and MPI	37
Running FieldView with MPI Parallel Servers	38
FieldView Parallel Performance Results	42
Obtaining Timing of Various Postprocessing Operations	45

Chapter 2

FieldView Tutorial Introduction	51
Prerequisites	51
Tutorial Data	52
Working Through FieldView Tutorials	54
Working with Aerospace Data	56
Working with Free Surface Flows	57
Working with Automotive Models	58
Working with Combustion Models	60
Working with Turbomachinery Data	62
Working with Transient Data	64
Working with Polymer Processing Data	66

Working with Biomedical Device Data	68
Working with Mixing Models	70
Working with Feature Extraction Techniques	71
Working with Heat Exchanger Models	73
Quick Pick	75
Quick Pick to Reduce Streamline Clutter	75
Restart Files	77
FVX Tutorial Scripts	77

Chapter 3

Basic Aerospace Tutorial.....	79
What You Will Learn	79
Step 1: Read the PLOT3D Tutorial Data	80
Step 2: Create Boundary Surfaces	81
Step 2: Create Boundary Surfaces (cont'd)	82
Step 3: Display Velocity Vectors	83
Step 4: Display the Pressure Coefficient	84
Step 5: Visualize the Vortex Core	85
Step 6: Create Streamlines	86
Step 7: Display the Mach Number	87
Step 8: Create a Scalar Function	88
Step 9: Create 2D Plots	89
Step 9: Create 2D Plots (cont'd)	90
Step 10: Create a Keyframe Animation	91

Chapter 4

Free Surface Flow Tutorial.....	95
What You Will Learn	95
Step 1: Read FLOW-3D® Data	96
Step 2: Visualize Transient Free Surface Flow	97
Step 3: Display Velocity Vectors on a Surface	98
Step 4: Create a Transient Flipbook Animation	99
Step 5: Create an Advanced Visualization	100

Chapter 5

Basic Automotive Tutorial	103
What You Will Learn	103
Step 1: Read in FLUENT-UNS Data	104
Step 2: Capture Vortex Formation with Velocity Vectors	105
Step 3: Display Vortex Formation with Streamlines	106
Step 4: Create a Flipbook Animation	107
Step 5: Create a 2D Plot Along a Surface	108
Step 5: Create a 2D Plot Along a Surface (cont'd)	109
Step 6: Integrate Pressure	110

Chapter 6

Basic Combustion Tutorial	111
What You Will Learn	111
Step 1: Create Wall Boundary Files	112
Step 2: Create the Burner Inlet and Tube Coil Walls	113
Step 3: Create Filled Contours on a Scalar Surface	114
Step 4: Visualize the Flame Envelope	115
Step 5: Visualize the Combustion Process	116
Step 6: Use Streamlines to Visualize the Flow Pattern	117
Step 7: Visualize the Gas Temperature	118

Chapter 7

Basic Turbomachinery Tutorial.....	119
What You Will Learn	119
Step 1: Working with Turbomachinery Data	120
Step 2: Use an Iso-Surface to Show Flow Blockage	121
Step 3: Streamline Seeding Using an Iso-Surface	122
Step 4: Visualize Flow Leakage	123
Step 5: Create Radial Pressure Contours	124

Chapter 8

Vortex Shedding Tutorial.....	125
What You Will Learn	125
Step 1: Read a Transient Dataset	125
Step 2: Perform a Transient Sweep	128
Step 3: Create Streamline Seeding Surfaces	129
Step 4: Create Streamlines	130
Step 5: Create Streaklines	131
Step 6: Import Streaklines and Improve your Animation	132
Step 7: Using Scripts and DataGuideTM	133

Chapter 9

Polymer Processing Tutorial	137
What You Will Learn	137
Step 1: Create Boundary Surfaces	138
Step 2: Visualize Back Flow	139
Step 3: View Axial Velocities	140
Step 4: Change the Scalar Display of the Cross Sections	141
Step 5: Integrate Concentration Variance of Each Cross Section ...	142
Step 6: Create a Keyframe Animation	143
Equations for Static Mixer	146
FieldView Equations	146

Chapter 10

Biomedical Tutorial	149
What You Will Learn	149
Step 1: Append Two Datasets	150
Step 2: Visualize Recirculation Zones	151
Step 3: Display the Shear on the Artery Wall	152
Step 4: Visualize Stress and Concentration Contours	153
Step 5: Calculate the Mass Balance	154
Step 6: Visualize the Drug Delivery	155
 Chapter 11	
Chemical Processing Tutorial	157
What You Will Learn	157
Step 1: Read in FV-UNS Data	158
Step 2: Analyze Flow around the Rotating Shafts	159
Step 3: Examining Mixing Patterns	160
Step 4: Use the Function Calculator to Compute the Flow Number	162
Equations for Chemical Processing	164
 Chapter 12	
Automotive Diesel Combustion Tutorial	165
What You Will Learn	165
Step 1: Create a Hidden Line View	166
Step 2: Viewing Velocity Vectors and Fuel Concentration	168
Step 3: Amplifying Contour Range and Animating Spray Results ...	170
Step 4: 2D Plots of Gasoline and O2 Concentration	172
Step 5: Display the Exhaust Mass Flow as a Function of Crank Angles	174
 Chapter 13	
Feature Extraction Tutorial	177
What You Will Learn	177
Step 1: Create Vortex Cores	178
Step 2: View Surface Flows	179
Step 3: Analyze Two Datasets Simultaneously	180
Step 4: Show Internal Flow Structure	181
Step 5: Create a Keyframe Animation	183
 Chapter 14	
Heat Exchanger Tutorial	185
What You Will Learn	185
Step 1: Display with Cylindrical Coordinates	186
Step 2: Automatically re-create surfaces from a different dataset ...	188
Step 3: Compare Datasets	190
Step 4: Use Partial Integration for Convective Flow Calculations ...	192
Step 5: Create a 2D Plot of Temperature	194

Step 5: Create a 2D Plot of Temperature (cont'd)	195
Step 6: Create a KeyFrame Animation using Multiple Datasets	197

Chapter 15

Streamline Seeding Tools.....	199
Spherical Seeding	200
How to Use Spherical Seeding	200
Spherical Seeding in the Feature Extraction Tutorial Dataset	201
Spherical Seeding in the Basic Aerospace Tutorial Dataset	204
Rectangular Seeding	208
How to Use Rectangular Seeding	208
Rectangular Seeding in the Basic Combustion Tutorial Dataset	209
Rectangular Seeding in the Chemical Processing Tutorial Dataset ..	213
Circular Seeding	215
How to Use Circular Seeding	216
Circular Seeding in the Basic Automotive Tutorial Dataset	217
Circular Seeding in the Biomedical Tutorial Dataset	220

Chapter 16

Working with XDBs Tutorial	225
What You Will Learn	225
Step 1: Create an XDB dataset	226
Step 2: Import the XDB dataset and display the results	227
Step 3: Using XDB datasets for Dataset Sampling	228
Step 3 (cont'd): Using XDB datasets for Dataset Sampling	229
Step 4: Performing Dataset Comparison with the Sampled Data	230
Step 4 (cont'd): Performing Dataset Comparison with the Sampled Data	231
Step 4 (cont'd): Performing Dataset Comparison with the Sampled Data	232

List of Figures

Figure 1:	Client-Server Operation on Remote Systems	3
Figure 2:	Remote Desktop Operation with Remote Graphics Software	4
Figure 3:	Remote Desktop Operation with Hardware	5
Figure 4:	Remote Desktop Operation with Software	6
Figure 5:	FV-UNS Panel with GRID PROCESSING	26
Figure 6:	Direct versus Client-Server Mode	28
Figure 7:	Client-Server Configurations	29
Figure 8:	Server Selection from the Data Input Menu	40
Figure 9:	Grid Subset Selection and Outline	41
Figure 10:	FieldView Parallel Scale-Up for Tetrahedral Data	43
Figure 11:	FieldView Parallel Scale-Up for Reading Data	43
Figure 12:	FieldView Parallel Scale-Up for Reading Data	44
Figure 13:	Partitioner enabled but set to OFF.	46
Figure 14:	Here, we see a case with its original grid list at left, and at right, the new grid list obtained by the Auto Partitioner. 47	
Figure 15:	Auto Partition Ghost Cells specified.	48
Figure 16:	An Iso Surface visualized after partitioning, with image at left showing use of no Ghost Cells, and 2 Ghost Cells at right. See how the gap artificially created by the Auto Partitioner got closed. 48	
Figure 17:	Quick Picking	75
Figure 18:	Streamlines Formation Below Engine Valve	76
Figure 19:	Deleting Streamlines	76

Chapter 1

1

Starting Field-View

Running FieldView

Following a successful installation:

LINUX users will run FieldView by typing `fv` in a console window. (You will need to ensure that you are either in the directory `FV_HOME/bin` or that you have added this directory to your path. If you have not done this, you will need to type the full path name where FieldView is installed.)

MAC users can simply click on the Application widget. However, to provide for FVX script debugging and the use of command line options, FieldView must be started in a mac terminal, as follows:

```
/Applications/FieldView-21.app/Contents/macos/fv <FV arguments>
```

If FVX debugging is not needed, `stderr` and `stdout` can be redirected normally. Alternatively, `stderr` and `stdout` can be redirected as follows:

```
open --stdout <path> --stderr <path> /Applications/FieldView-21.app \
--args <FV arguments>
```

!

WINDOWS users will launch **FieldView** from the Program group, navigating to the **Field-View 2023** folder and clicking on **FieldView 2023** to start the program. To run **FieldView** from a WINDOWS command prompt or a Cygwin shell, type `fv.bat` followed by any desired command line arguments. (You will need to ensure that you are either in the folder `FV_HOME/bin` or that you have added this directory to your path. If you have not done this, you will need to type the full path to `fv.bat`.)

Special Notes for MAC Users



The command key on the MAC replaces the [Ctrl] key for operations to set the center of view and rotation, all picking (such as probing, streamline seeding and iso-surface cutting plane creation) and 2D plotting. Middle mouse button operations are enabled using the left mouse button while holding down the alt/option key. In addition, the MAC port has the following limitations:

- the **FVX** `make_panel` and `graph` commands are currently not supported
- the `acusim`, `cgns` and `flow3d` animation & restart direct readers are not supported since they have not been ported to the MAC platform by their respective development teams

Running Two Versions of **FieldView** Simultaneously

When **FieldView** is started, it uses a default port number of 12345. Each instance of **FieldView** must use a unique port number. You can run two or more versions of **FieldView** on a single machine, assuming your licenses support this, by specifying an explicit port number (where `port_number` is any port that is not in use by another process) as follows:

```
fv -port port_number
```

It is also possible to start FieldView by specifying a port number, range of port numbers, where FieldView will use the first open port in that range, or **-port none**:

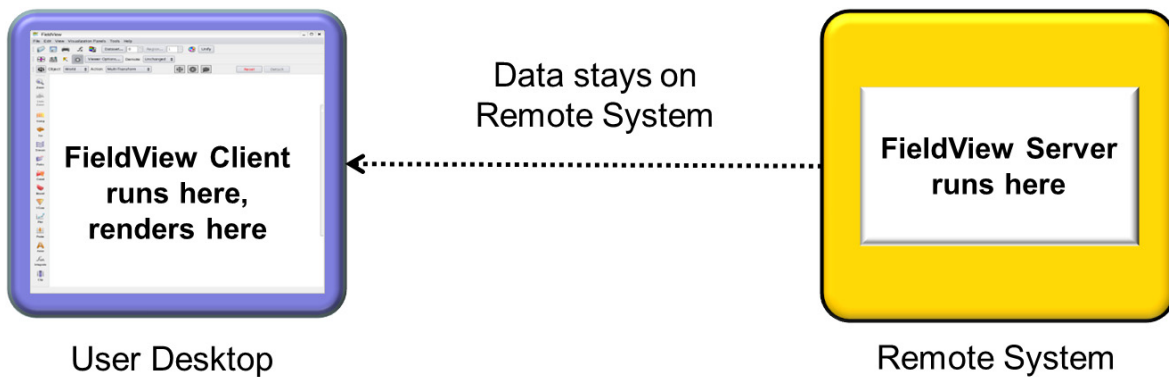
```
fv -port 21000:22000
```

However, a the -port none option will prohibit use of FieldView Servers.

FieldView also provides the environment variable `FV_DEFAULT_PORT` for this setting.

Remote FieldView Operation

FieldView can be run remotely, thereby avoiding the need to move data from your remote system to your local desktop system. **FieldView** visualization performance while running in Client-Server (see [Running Client-Server Mode](#)) vs. running with a remote display (with or without the `-srg/-hrg` options) is dependent on your specific environment, whether remote graphics software is being used, and the content of your visualization.



Start **FieldView** normally, no command line switches

Figure 1 Client-Server Operation on Remote Systems

With releases prior to **FieldView** 13, it was possible to use an '-x' mode to run **FieldView** on a remote display. A requirement for this mode of operation is that the user desktop needed to have an X server running. Also, only software rendering was used at the source or remote desktop. The table below describes the different interactive configurations supported for **FieldView** versions prior to **FieldView** 13, using '-x' mode.

User Desktop	Transport	Source Machines(s)										
X Server (software rendering at source)	X Windows	Single FieldView Process**										
		run in -X mode										
		FieldView Client**										
		run in -X mode + Server (handles big data)										
		FieldView Client**										
		run in -X mode , Controller & FieldView Worker Servers (Parallel Operation)										
		<table><tr><th>Head Node</th><th>Cluster</th></tr><tr><td>FieldView Client**</td><td>FieldView Worker</td></tr><tr><td>run in -X mode</td><td>Servers (Parallel</td></tr><tr><td>FieldView Controller</td><td>Operation)</td></tr><tr><td>Server</td><td></td></tr></table>	Head Node	Cluster	FieldView Client**	FieldView Worker	run in -X mode	Servers (Parallel	FieldView Controller	Operation)	Server	
Head Node	Cluster											
FieldView Client**	FieldView Worker											
run in -X mode	Servers (Parallel											
FieldView Controller	Operation)											
Server												

Note: ** above indicates the process that needs to communicate with the License Manager

With **FieldView** 13, permitting X Windows transport of the graphics window is no longer possible, and as a result the remote desktop operation feature based on '-x' operation is unavailable. The high performance rendering changes delivered with **FieldView** 13 require a robust, GL context for the graphics window. Notably however, **FieldView** does

run well with Remote Graphics Software applications such as HP RGS, as illustrated in the figure below.



Start **FieldView** normally, no command line switches

Figure 2 Remote Desktop Operation with Remote Graphics Software

With **FieldView** 13.1, a mode of operation equivalent to '-x' mode has been provided. This restores the ability to run **FieldView** from a remote desktop, displaying to a user desktop. The table below describes the current interactive configurations for running **FieldView** on a remote system.

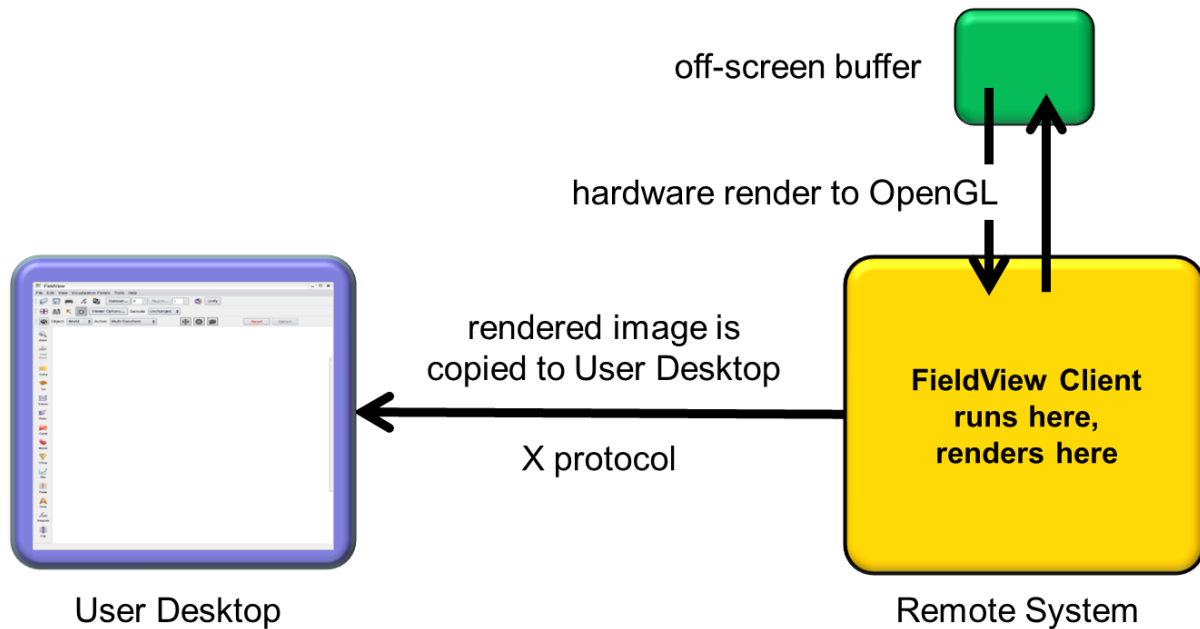
User Desktop	Transport	Source Machines(s)	
FieldView Client ** (Local OpenGL)	RPC/Sockets	FieldView Server (handles big data)	
		FieldView Master Server & FieldView Slave Servers (Parallel Operation)	
		<div>Head Node</div> FieldView Master Server	<div>Cluster</div> FieldView Slave Servers (Parallel Operation)
Remote Graphics Software Applications (HP RGS, VNC for Windows, etc)	Remote Graphics Software Application	Single FieldView Process**	
		FieldView Client** (plus any configuration in the tables above)	

Note: ** above indicates the process that needs to communicate with the License Manager

X protocols are still needed to run **FieldView** in this way. There are two implementations for remote operation which apply only to non-Windows **FieldView** Client. One is based on hardware rendering being performed on the remote system; the other is based on software rendering.

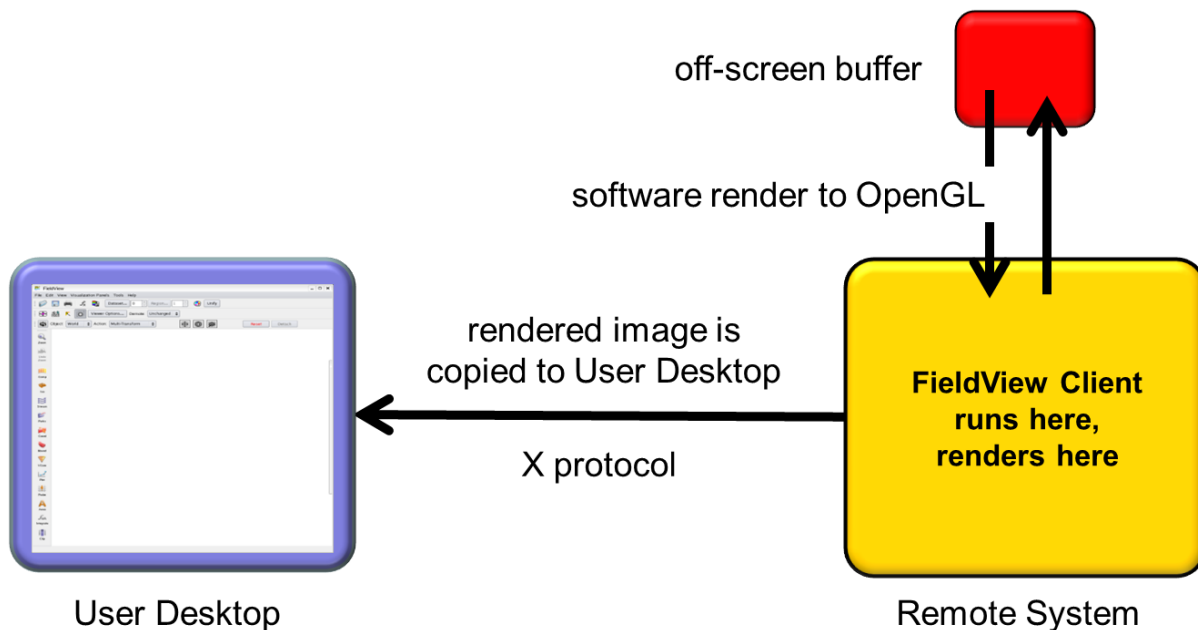
NOTE that attempting these command line options with the Windows **FieldView** will result in the following message:

This rendering option (-hrg) is not supported on this platform.



Start **FieldView** using `-hrg` (Hardware Remote Graphics)

Figure 3 Remote Desktop Operation with Hardware



Start **FieldView** using `-srg` (Software Remote Graphics)

Figure 4 Remote Desktop Operation with Software

Note that the hardware based implementation delivers higher performance than the software option. If software rendering is necessary, using a Vertices Display Type instead of Smooth shading is among the strategies for improving performance.

When starting **FieldView** on a remote system, it is likely that you will need to set a command line argument. In the case that you cannot get a display which **FieldView** can use, you may see the following message in the console window:

```
Unable to run FieldView on your remote system
Starting FieldView with the -hrg switch may resolve this problem.
```

To run **FieldView** remotely using Hardware remote graphics, use:

```
fv -hrg
```

Note that you may need to set a display other than the default of `:0.0`. To do this, you can add an argument as follows, to run on a display setting of `:1.0`:

```
fv -hrg :1.0
```

It is possible that you will be unable to run using the hardware remote graphics mode on your remote system. This is most likely due to the fact that you do not have an X session

running on your remote system. In this case, you will see the following message in your console window:

```
Unable to allocate pBuffer for rendering
Starting FieldView with the -srg switch may resolve this problem.
```

Running multiple **FieldView** Client processes with command line option `-hrg` on a system with a single graphics board can run out of resources on that device, yielding black frames or small images. Small images seem to result when X server tools such as VNC are used. This problem is caused by running out of memory resources on the graphics board. Running fewer concurrent jobs or adding memory to the graphics board are the known workarounds.

To run **FieldView** remotely using Software remote graphics, use:

```
fv -srg
```

Please note that the command line switches `-hrg`, `-srg` and `-batch` cannot be used together. If they are used together, only the last one will prevail.

Please note that the following command line switches are no longer supported: `-ogl`, `-gl`, `-sb`, `-xgl`, `-pseudo`, `-x`, `-x8`, `-X8`. If you try to run **FieldView** with one of these switches, **FieldView** will not start, and you will see the following message in the console window:

```
This rendering option (<switch name>) is no longer supported.
```

Understanding the Console Window

The console window is used to send messages concerning the normal, and sometimes unsuccessful, operation of **FieldView**. For LINUX/MAC users, the console window refers to the window in which you started **FieldView** by typing `fv` (along with any appropriate command line arguments). For WINDOWS users, the console window is a smaller window, separate from the main **FieldView** interface. A great deal of information is typically printed here. However, these console window messages can often be very helpful in diagnosing problems. Some of these messages and their relevance are highlighted here. For instance, it is often useful to know the current working directory:

```
Current working directory is /home/my_data/cases
```

By default, **FieldView** will remember your last current working directory, and this may be confusing if you have navigated to a different directory since your last session. This behavior can be over-ridden using either of the environment variables, `FV_OLD_BROWSER_BEHAVIOR` or `FV_NO_PREFERENCES`. Checking the console window helps to confirm which directory **FieldView** considers to be 'current'.

You may encounter difficulties when running for the first time with a basic server or parallel server. **FieldView** will echo out all relevant information concerning the configuration of the server it is trying to run:

```
Reading dataset with Server: local
Server current working directory:
/opt/apps/fieldview/fv2023/fv/bin
Server configuration file contents:
ServerType: standard
```

FieldView, by default, is configured to run with high performance graphics hardware. For a normal start-up, you should see some messages confirming the use of FBOs (Frame Buffered Objects):

```
FIELDVIEW using FBO rendering
FBO usage:          7.03 MB = 640 x 480 x 6 x 4
```

On systems with lesser graphics capacity, you may need to set the environment variable, `FV_RF_GRAPHICS`. With this set, the FBO messages will be replaced by:

```
FIELDVIEW using FB rendering
```

Remote operation can be particularly difficult to understand, and problems are not always straightforward to diagnose. When running remotely, you should see something like:

```
Display is set to localhost:10.0
Remote rendering (-hrg) switch enabled
FieldView is running on a remote display using accelerated rendering
```

These messages should help to confirm your expectations of what resources **FieldView** is using to run.

The console window may help provide you with sufficient information to resolve problems encountered when running **FieldView**. Additionally, the information will be helpful to our Customer Care team in getting you up and running again.

Command Line Switches

FieldView has several switches (or arguments) that control various aspects of program operation. It is possible to use multiple command line switches when starting **FieldView**. In the event that switches may set conflicting behavior, such as setting a remote operation switch followed by the batch switch (`-hrg -batch`), the last switch will be used. The switches are summarized in the following table and further explained elsewhere in the text.

Argument	Description
-batch [-software_render :displaynum :displaynum.screen]	run in batch mode ; Optional arguments shown here within [] specify Software Render (Mesa) or Hardware Render DISPLAY target.
-fvbN	use batch-only license, batch mode is forced N (optional) refers to the number of parallel processes which can be run in batch-only mode and can be set to 8, 16, 32, 64 or 256. A batch-only license, consistent with (optional) N, is needed for this switch to work correctly. (1)
-f name	run a restart when starting FieldView
-s name	run a script when starting FieldView
-fvx name	run an FVX program when starting FieldView
-pfvN	use parallel license N refers to the number of parallel processes which can be run in parallel (not including the controller process) and can be set to 8, 16, 32, 64 or 128. A parallel license, consistent with N, is needed for this switch to work correctly. (1)
-port [port-num port:range]	run a python program when starting FieldView
-python name	run a python program when starting FieldView
-hrg display	run FieldView on a non-Windows remote system using Hardware Remote Graphics, with an optional display specification
-srg display	run FieldView on a non-Windows remote system using Software Remote Graphics, with an optional display specification
-pN	DataGuide™ switches: 0 : <i>only Grid file</i> 1 : <i>both, function subset</i> 2 : <i>both, all functions</i>
-fc	use PLOT3D/Fast mouse controls
-create_exterior_fvbnd	Generate FVBND file (must be used with -pN)
-create_wall_fvbnd	Generate FVBND file (must be used with -pN)

-gamma <i>value</i>	set PLOT3D gamma (default=1.4)
-gasconstant <i>value</i>	set PLOT3D gas constant, R (default=1.0)
-silent	suppress all warning pop-ups
-size=nnn×mmm	set the graphics window size
-port <i>port_number</i>	set the port number to use for Client-Server or Parallel operation
-software_render	force software rendering (must be used with -batch)

(1) Only applies to **FieldView**

Batch Mode

FieldView can be run in batch mode without displaying the graphical user interface. **Batch** mode is enabled by starting **FieldView** with the **-batch** argument. It must be used in conjunction with an **FVX** program (**-fvx name**), a Python program (**-python name**), an automatic script restart (**-s name**) or a restart (**-f name**). **DataGuide™** processing can also be used.

When **FieldView** is run in batch mode from the command line using the **-batch** switch, it does not require an X-server to run. This is a major change in how **FieldView** runs in batch compared with previous releases.

To run a script in batch, use:

```
fv -batch -s script_name
```

To run a restart in batch, and generate a **DataGuide™** file at the same time, use:

```
fv -batch -pN -f restart_name
```

Batch mode, when combined with script commands such as PROBE, INTEGRATE and EXPORT, will allow you to analyze many datasets in succession, examine very large and/or transient datasets non-interactively, and run **FieldView** overnight.



Note: Scripts which do not have an EXIT command will exit at the end of the script when **batch** mode is enabled. Script errors which normally give the Continue/Cancel option force Cancel, and exit the script in **batch** mode.

Hardware Render for Batch

When batch mode is requested by running **FieldView** with the **-batch** switch, you will see the following message in your console window:

```
Attempting to use hardware for batch rendering.
```

FieldView will attempt to run using accelerated off-screen rendering via a pbuffer. If the pbuffer initialization fails, then you will see the following message:

```
Batch mode initialization failed.  Retrying with software rendering.
```

When this happens, **FieldView** will run the batch job using software rendering. There is no need to restart **FieldView**; the batch job is re-run automatically.

To provide the greatest flexibility, the `-batch` switch now supports an optional command line argument to provide further control for pbuffer allocation:

```
:displaynum    OR    :displaynum.screennum
```

This argument specifies the display on which the pbuffer will be allocated. This feature behaves similarly to the display setting when the `-hrg` (Hardware Remote Graphics) option is used.

It is still possible to force **FieldView** to use software rendering. This is done by setting the `-software_render` command line switch in conjunction with `-batch` command line switch. Please note that this switch only affects `-batch`; it has no effect on `-hrg`/`-srg` operation.

Important notes and limitations

Hardware rendering is never used for Batch Pack licenses. If a `-fvb` switch is used on the command line to start a batch job, software rendering will be used.

Hardware rendering is not available for MAC platforms due to an unavailability of pbuffer support for Cocoa.

Using Batch-only licensing

FieldView can run using a reduced cost batch license, which only permits batch operation. When running with this type of license, the **FieldView** interface will not be launched. The purpose of this type of license is to let you run many **FieldView** batch jobs without having to use your interactive licenses.

To run a batch job using a batch-only license, use one of the command line options `[-fvb | -fvb8 | -fvb16 | -fvb32 | -fvb64 | -hpcf vb]`. (This forces **FieldView** to run in batch mode, making the `-batch` command line option unnecessary.)

```
fv -fvb -fvx fvx_name
```

NOTE: If no reduced cost batch licenses are available, **FieldView** will try to check out a standard license and run in batch mode. However, when using a batch-only command

line option (as opposed to the `-batch` option), your batch job will not use hardware accelerated rendering, even if graphics hardware is available.

Running Restarts from the command line

A **FieldView** Restart can be run at start-up using the following command line argument:

```
fv -f restart_name
```

On startup, **FieldView** looks for a `-f` name argument on the command line. If this argument is present, **FieldView** will automatically read any files that begin with `restart_name` and have a standard **FieldView** file extension (such as `restart_name.map` for a Colormap restart file, or `restart_name.iso` for an Iso-surface restart file). Note that name can include slashes to access directories other than the current directory, however file extensions are *not* allowed.

Preference Restart

If no `-f` argument is present, **FieldView** will automatically read in any files in the current directory that begin with `fv` and have a standard **FieldView** file extension. If no `fv.*` files exist in the current directory, **FieldView** will read in any such files that exist in the directory `FV_HOME/data`. This allows for preference files to be read in automatically upon startup.

An automatic restart of your user preference settings can be created with the Preference Restart option on the Restart Files menu. Restart files are described in [Chapter 5](#) of the **Reference Manual**.

Running Scripts from the command line

A **FieldView** script can be run at start-up using the following command line argument:

```
fv -s script_name
```

where `script_name` is the name of the script file. This is often used in conjunction with Batch mode (see above). Script files are described in [Chapter 5](#) of the **Reference Manual**.

Running **FVX** from the command line

An **FVX** script can be run at start-up using the following command line argument:

```
fv -fvx fv_name
```

where `fv_name` is the name of the **FVX** program file. This is often used in conjunction with Batch mode (see above). **FVX** programming is described in [Chapter 4](#) of the **Reference Manual**.

Running Python from the command line

A python script can be run at start-up using the following command line argument::

```
fv -python python_name
```

where name is the name of the python program file. This is often used in conjunction with Batch mode (see above). **Python-enabled FVX** programming is described in [Chapter 4](#) of the **Reference Manual**.

DataGuide™

DataGuide™ lets you create specially formatted files (based on grid and results files) that will *substantially* decrease read in time. The time savings for very large transient datasets, for example, can be extraordinary. **DataGuide™** licensing is included to both *create* and also to *use* **DataGuide™** files. Command-line switches are needed to *create* **DataGuide™** files. Once the files are created, if they reside in the same directory as the grid and results file, they will be *automatically* used by **FieldView**. The data read-in process is no different for you. Note that this feature cannot be used if your data has been read with Parallel FieldView, unless read in Partitioned File (PFPR) format. If tried, FieldView will issue a corresponding message.



Note: **DataGuide™** files are created based on your Grid Processing settings - "Less", "Balanced" or "More". These are controlled interactively (on the Data Input Panel) or in restarts (.dat) and **FVX**. (For more details, see the [GRID PROCESSING](#) section of [Chapter 1](#) of the **Reference Manual**.)

To make **DataGuide™** pre-processing files:

Start **FieldView** with an appropriate command line switch to create grid and/or results **DataGuide™** files. We strongly recommend creating **DataGuide™** files in **batch** mode operation, however **DataGuide™** files can be created during an interactive session. As soon as a dataset is read, either a **DataGuide™** grid pre-processing file, or a **DataGuide™** grid and a results pre-processing file are created. If you have a transient dataset, perform a transient sweep to create the pre-processing files for each time step.

To use **DataGuide™** pre-processing files:

After having created the **DataGuide™** pre-processing files, start a new **FieldView** session without the command line switches. The **DataGuide™** pre-processing files will be automatically read when the dataset files they are based on are read into **FieldView**.

DataGuide™ Grid pre-processing

DataGuide™ Grid pre-processing causes **FieldView** to create a file based on the filename of the grid file having the extension: .fvpre. Although requiring additional disk

space (the original grid file must still remain), having **DataGuide™** grid files will *substantially* decrease read in times and access times for data. The performance increase will be greater for structured grids than for unstructured grids.



Note: A **DataGuide™** switch (-p0, -p1, or -p2) is required when using the (Structured Dataset options) `-create_exterior_fvbnd` or `-create_wall_fvbnd` command, but is unnecessary for the equivalent interactive options Tools > Create Exterior Boundary File, or Tools > Create Wall Boundary File. For a description of these two utilities, see [Create Exterior Bnd File](#) and [Create Wall Bnd File](#) in [Chapter 14](#) of **Working with FieldView**. For a description of the Structured Boundary file format, see [Appendix H](#) of the **Reference Manual**.

DataGuide™ Results pre-processing

DataGuide™ Results pre-processing causes **FieldView** to create a file based on the file-name of the results file having the extension: `.fvres`. (For solvers that combine grid and results data in a single file, the name would be based on the single file name). **FieldView** will calculate the minimum and maximum values of solver variables (and PLOT3D functions) and store them in this file. These files are readable text and also report the IJK and XYZ values of the minima and maxima per function / per grid. For scalar functions **FieldView** will also report where the minimum and maximum values occur, I/J/K for structured grids and X/Y/Z for structured and unstructured grids. Vector functions will not show the min/max locations except by their component scalar function listings.

An example listing for a scalar function will look like this:

```
Function = "Cp [PLOT3D]"
grid = 1      fid = 114      min = -0.104507      max = 2.13695
  min at I/J/K = 27          13          19
        X/Y/Z = 0.78778      0.624087      0.407535
  max at I/J/K = 2          4          24
        X/Y/Z = -0.00756693  0.0189921      1.13551
```

An example listing for a vector function will look like this:

```
Function = "Velocity Vectors [PLOT3D]"
grid = 1      fid = 200
u-min = -2.05706      u-max = 3.12293
v-min = -0.348877      v-max = 1.88989
w-min = -2.00529      w-max = 1.13023
```

(The `fid` value is for internal **FieldView** use only. It should not be edited.)

Implementation

There are three different command line switches depending on the amount of **DataGuide™** processing you want to do:

-p0 means build *only* the grid **DataGuide™** file
 -p1 means build both the grid and results **DataGuide™** files, and for the case of PLOT3D, use only a subset of functions so as to save time. These functions are:

```
"Density (Q1)"
"Cp [PLOT3D]"
"Temperature [PLOT3D]"
"Velocity Magnitude [PLOT3D]"
"Mach number [PLOT3D]"
"x-momentum (Q2)"
"y-momentum (Q3)"
"z-momentum (Q4)"
"Stag. energy (Q5)"
"Velocity Vectors [PLOT3D]"
"Momentum Vectors [PLOT3D]"
```

-p2 means build both the grid and results **DataGuide™** files and for the case of PLOT3D, use *all* functions for results processing.

If more than one of the above switches appears on the command line, **FieldView** will perform the one that does the least. If **FieldView** is started with a command line switch to create a **DataGuide™** file, and the **DataGuide™** file already exists, the files will be overwritten.

DataGuide™ processing can be used in conjunction with a Script or a Restart, or by itself:

```
fv -p2
```

```
fv -p2 -f restart_name (FieldView will exit upon completion)
```

```
fv -p2 -s script_name (FieldView will exit upon completion)
```

```
fv -p2 -fvx script_name
```



Note: Grid and Results **DataGuide™** processing will not function for transient data where the grid and results are all in a single file (e.g. STAR-CD, CFX).

Other information:

FieldView will print warnings or errors when encountering **DataGuide™** files which may not be optimal or are unreadable with the current **FieldView** version. For example, if grid sampling or subsetting doesn't match, **FieldView** will print:

```
Number of cells in DataGuide file does not match grid data.
```

For example, reading a preprocessed case where your current settings differ, **FieldView** will print:

```
The Grid Processing setting was ignored because a DataGuide file
was found and used.
```

PLOT3D/Fast Viewing Controls

When the “-fc” command line option is used when running **FieldView**, the transform specification of the mouse buttons is changed. Instead of the normal **FieldView** definitions of the three default mouse operations, FAST-like transforms are mapped to the mouse buttons. In addition, the default demotion mode will be None and the default tracking will change from Track to Running.

Modifying Constants for PLOT3D Functions

In order to change the internal values of the gas constant (**R**) and gamma (γ), command line switches will have to be used. The command line switches are:

```
-gasconstant  value
```

The possible values of the gas constant are values strictly greater than 0. The normal value used in the PLOT3D functions is $R = 1$. This assumes a perfect gas. Whether the value of the gas constant you enter is required to be normalized or not will depend on your normalization of these quantities in your solver.

```
-gamma  value
```

The value of gamma has to be strictly greater than 1.0. The normal value used in the PLOT3D functions is $\gamma = 1.4$.

If either the gas constant (**R**) or gamma (γ) is supplied by the user the value(s) will be echoed in the console window.



Note: The PLOT3D derived functions will still assume that “rho-inf” (freestream density) is 1, and the “c-inf” (free stream speed of sound) is 1, regardless of the values of the gas constant (**R**) and gamma (γ).

Effects on DataGuide™ Files

1. The `fvres` files have a format that lists the gas constant (**R**) and gamma (γ) at the top of the file.

2. `fvres` files in the standard format will be ignored unless the current values for the gas constant (**R**) and gamma (γ), are set to their defaults. A warning message will indicate a mismatch in these values.
3. `fvres` files in the modified format will be ignored if the values of the gas constant (**R**) and gamma (γ) in the **fvres** file *do not* match the values currently set on the command line. This will produce a warning message.

Graphics Layout Size

The graphics window size can be controlled during both interactive and batch mode operation of **FieldView**. The default window size while running **FieldView** interactively is discussed in the [Graphics Layout Size](#) section of **Working with FieldView**, and can be controlled with Graphics Layout Size from the Tools or View menu, or the `-size` command line option. The default window size when running in batch mode (with the `-batch` command line option) is 640x480.

You can control the image size by using the following command line argument:

```
fv -size=WIDTHxHEIGHT
```

where `WIDTH` and `HEIGHT` are measured in pixels. For example, to set the image portion of the graphics window to 640x480, use the following command to run **FieldView**:

```
fv -size=640x480
```

This command line argument works in conjunction with all of the other **FieldView** arguments, for restarts, scripts, batch mode, etc.

FieldView Environment Variables

Environment variables can give you greater control over how **FieldView** works following a standard installation. In most cases, an environment variable is activated by setting it to a non-zero value. Setting environment variables will depend on your operating system. For LINUX users, it will also depend on the shell you are using.

In general, an environment variable applicable to the FieldView Server(s), which is set at the FieldView Client, even if running on a remote system, will be applied to the server(s) unless specifically noted. Alternatively, environment variables applicable to the FieldView Server(s) can be set when running **fvrnsrv** to launch the server. Note: Use care in setting environment variables on Windows systems, as trailing spaces will not be discarded!

Variable	Definition
FVEXP_NUM_LOCAL_PARALLEL	Adjusts the number of worker processes used in Local Parallel mode. Also impacts the amount of HWUs being drawn. (2)
FV_2D_TO_3D	Use this to extrude a 2D dataset to 3D, enabling FieldView to perform volume-based 2D line plotting and point probe. Only applies to data in the binary FieldView Unstructured format (FVUNS).
FV_ALT_LOOKUP	Use this if you experience slow performance when attempting to move a surface with many faces or polygons (generally more than 1 million). For large datasets, or datasets with many regions, we strongly recommend that you set this environment variable.
FV_ARB_POLY	When set, FieldView will tetrahedralize arbitrary polyhedra over-riding the default behavior to handle these elements natively. Use FV_TET_CONV to apply a quick scheme for tetrahedralization.
FV_ARB_POLY_FULL_CHECK	When set, reverts to the more extensive checking of arbitrary polyhedron cells done prior to FieldView 17 and prints a console warning the first time any data with arbitrary polyhedron cells is read. If you are writing code to export FV-UNS files, setting this while testing your export is recommended.
FV_DATA_CHECK	Use this to check for invalid floating-point values (NaN's) in PLOT3D and FV-UNS data. Infinities are not checked for. This will increase memory usage and slow down read in times. See also FV_NO_DATA_CHECK .
FV_DEBUG	Set this to obtain additional diagnostic information, printed to the console window, when running FieldView .

FV_DEBUG_AUTOSTART	Set this to see the exact server command line that FieldView is attempting to start.
FV_DEBUG_STARTUP	Helps debug FieldView problems with clients and servers on non-Windows systems that are caused by limited system resources. See Additional Comments Regarding FieldView Servers in the FieldView Installation Guide for a full description.
FV_DEFAULT_PORT	Use this to set the port number that FieldView uses for Client-Server or Parallel operation. Can take a single integer, or a range <port>:<port>.
FV_DPI	Set this to specify the “dots per inch” for PostScript, EPS and EMF rendering. The default value is 120. Note: The FV_DPI environment variable only affects EPS/PS GRAPHICS window printed files. The environment variable has no effect on images saved from the Plot window since they contain only vector graphics.
FV_GUI_FONT_SIZE	Set this to a number between 50 and 200 to change the font size of the GUI. A setting of 50 will scale the fonts by 50%, 200 will scale the fonts by 200% (or a factor of 2X)
FV_HOME	This variable is set and used by the FieldView start-up script. Changing this environment variable is NOT recommended.
FV_HQ_AVI	Set this environment variable to any value to use the high quality option when saving AVI files on Windows systems.
FV_IGNORE_WALLS	If streamlines appear to arbitrarily stop within a domain, it may be the case that internal walls within the dataset are being incorrectly flagged as surfaces which will not let streamlines pass. Setting this environment variable ignores internal walls for streamline calculations.
FV_MP4_COMPRESSION FACTOR	This environment variable specifies the Constant Rate Factor (CRF) quality adjustment for the H.264 codec when exporting flipbooks in MP4 format and may be set to an integer value from 0 to 51. A value of 0 will produce a lossless compression of the original image content at the expense of a large file size. A value of 51 will produce a highly compressed and probably low quality rendition of the original image content but with a drastically smaller file size. A default value of 23 has been chosen to be a good balance of fidelity vs. file size. A "sane" range of 18 for the highest quality and 28 for the smallest file size is recommended.

FV_MULTISAMPLE_ENABLE	This variable turns on full screen hardware anti-aliasing in the graphics and 2D plot window. This has no effect on hardcopy output.
FV_NO_BACKUP_RESTART	To avoid losing work, a Complete Restart is saved to your home directory when exiting Fieldview . To turn this default behavior off, set this environment variable.
FV_NO_BNDRY_FILE	This variable turns off creation of .fvbnd files by the WIND reader. For more details see Appendix H (section on the NPARC/WIND reader) of the Reference Manual .
FV_NO_DATA_CHECK	Use this to skip checks on element and boundary node IDs in PLOT3D and FV-UNS files. See also FV_DATA_CHECK .
FV_NO_FVX_RESTART	Use this environment variable to turn OFF the automatic creation of an FVX file when a complete or current RESTART is saved.
FV_NO_GRID_JUMP	This variable turns off streamline grid jumping, except for grid jumping caused by negative IBlanks.
FV_NO_OUTLINE	By default, FieldView computes and draws outlines when data is read. To expedite data reads and turn off outline drawing, set this environment variable to any value.
FV_NO_PREFERENCES	By default, all GUI layout customization and the locations of the last directory browsed for all file selectors are saved. To completely disable this default behavior, set this environment variable.
FV_NO_SCONFIG_ECHO	By default, detailed information regarding servers (client-server or parallel) is printed to the console window. Set this environment variable to disable this default behavior. (1)
FV_NO_STREAK	By default, FieldView will calculate streaklines from streamline rakes during a transient sweep operation. Use this environment variable to disable this behavior. When set, a transient sweep will produce streamlines instead of streaklines.
FV_NUM_THREADS	Thread usage is normally limited by the OMP_NUM_THREADS environment variable on the system. FV_NUM_THREADS can be set to further limit the number of threads used by FieldView.
FV_OLD_BROWSER_BEHAVIOR	By default, the locations of the last directory browsed for all file selectors are saved. This may cause confusion (particularly for LINUX users) since file browsers may not start in your current working directory. To override this default behavior, use this environment variable.

FV_PDF_VIEWER	For LINUX users, setting this variable to point to a PDF Viewer will allow you to override the Acrobat PDF Viewer default.
FV_PLUGINS	By default, FieldView expects plug-in readers to be located in the FV_HOME/bin/plugins directory. Use this to specify a different location for your reader source and libs for plug-in readers. (1)
FV_PRE17_ARB_POLYS	When set to any value, reverts to the previous behavior for reading arbitrary polyhedron cells and prints a console warning the first time any data with arbitrary polyhedron cells is read.
FV_PRINT_DIR	This variable specifies a temporary directory for temporary files created by the Print menu.
FV_PRINT_NO_AA	Use this to disable anti-aliasing for postscript and emf image output.
FV_PROBE_PERFORMANCE	Setting this value to an integer number between 1 and 6 will increase the performance of all point probe operations at the cost of increased memory and longer read-in times. The internal default value for this setting depends on the amount of RAM on your system.
FV_PROBE_SAVE_MEM	Use this to lower the memory requirements needed to read data at the cost of decreased performance.
FV_RF_GRAPHICS	Set this environment variable if you encounter graphical problems or see error messages in the console window regarding OpenGL problems while running FieldView . This environment variable is designed to allow FieldView to run on systems with reduced functionality graphics support by disabling layered transparency and anti-aliasing.
FV_SCREEN_GRAB	On some systems with reduced graphics functionality, this variable may be needed to properly save images of the graphics or plot window to a file.

FV_SELECT_FUNCTIONS_ONLY	<p>This will limit results preprocessing of PLOT3D Q files to this select set of “critical” functions:</p> <ul style="list-style-type: none"> “Density (Q1)” “Cp [PLOT3D]” “Temperature [PLOT3D]” “Velocity Magnitude [PLOT3D]” “Mach number [PLOT3D]” “x-momentum (Q2)” “y-momentum (Q3)” “z-momentum (Q4)” “Stag. energy (Q5)” “Velocity Vectors [PLOT3D]” “Momentum Vectors [PLOT3D]”
FV_SERVER_CONFIG_DIR	<p>By default, FieldView looks for the server configuration files needed to run in Client-Server or Parallel mode in FV_HOME/sconfig. Use this environment variable to define the location of your server config files. This setting can be uniquely defined for each user. (1)</p>
FV_SHOCK_MIN_NODES	<p>Use this to set the minimum number of nodes in a connected shock region. The <i>default</i> is 4.</p>
FV_SHOCK_PRESSURE_CHANGE	<p>The minimum pressure change in a connected shock region is defined as:</p> $\max(\text{pressure}) - \min(\text{pressure})$ <p>Use this environment variable, expressed as a fraction of the total pressure range for the entire dataset, i.e., (shock pressure range) / (dataset pressure range) to customize how shock surfaces are calculated. The <i>default</i> is 0.01.</p>
FV_SHOW_WALLS	<p>If you see streamlines arbitrarily stopping in a dataset, it may be possible that there are internal walls blocking them. By setting this variable, an additional boundary type will be generated which can be used to show all walls in a dataset which streamlines are not allowed to pass through. See FV_IGNORE_WALLS to calculate streamlines, ignoring any walls which are marked by the solver to block streamlines.</p>
FV_SINGLE_FILE_STREAKLINES	<p>By default, FieldView uses the PARTICLE SET format to export one particle path per timestep when saving a transient sweep. Set this environment variable to any value to disable this default behavior and use the STREAKLINE format to export particle paths to a single file.</p>

FV_SSLINES_EXPORT	By default, surface streamlines are sampled at each point to acquire the current scalar, velocity, threshold and iso-surface register values. To reduce the time required to calculate surface streamlines, this sampling step can be disabled by setting this environment variable. When this environment variable is set, the current scalar, velocity, threshold and iso-surface values will not be written to the exported file.
FV_ST_STREAMLINES	Forces OFF multithreading for Streamlines
FV_ST_FORMULA	Forces OFF multithreading for Formula Calculation
FV_ST_VCORE	Forces OFF multithreading for VortexCore and SurfaceFlow
FV_STEREO_ENABLE	Set this in the system settings on your computer to enable stereo.
FV_TET_CONV	When set, FieldView will tetrahedralize arbitrary polyhedra using a quick scheme, over-riding the default behavior to handle these elements natively. Use FV_ARB_POLY to apply a more robust scheme for tetrahedralization.
FV_TRANSP_LAYERS	Use this to set the number of layers used for transparent surfaces. The default is 6; this value can range from 2 to 8. If you are running out of memory at the default level of 6, or if you wish to reduce the time it takes to render transparent surfaces, reducing this value may help.
FV_USE_FV13_XDB_FORMAT	Set this to create XDB files using the older XDB file format, which is compatible with the FieldView Advanced Technology Viewer.
FV_USE_FV14_GL_MODE	<p>FieldView will use Vertex Buffer Objects (VBOs) by default for rendering. This method will allow a significant decrease in memory required for rendering a given scene, while maintaining a similar level of performance, compared to the default method used in FieldView 14 (display list).</p> <p>The environment variable FV_USE_FV14_GL_MODE can be set to any value to revert to the GL mode used in FieldView 14. FieldView will then require more memory to render the same scene, but it may also perform faster on some older graphic cards or using older graphic drivers.</p> <p>Both GL modes can be used in conjunction with FieldView's various graphic modes:</p> <ul style="list-style-type: none"> - Local rendering - Remote graphics with -hrg or -srg - Batch with hardware or software rendering - Reduced functionality graphics with FV_RF_GRAPHICS

FV_USE_FV16_FONT	The GUI will use the user's default system font, including point size, as of FieldView 17. To revert to the previous behavior, set this environment variable.
FV_USE_LONGEST_PATH	The length of streamline filaments are based on the average duration of all streamlines in a rake. Use this to override the filament length calculation by using the longest duration streamline in a given rake.
FV_USER_HELP_ENTRY	If FV_USER_HELP_URL is set, replaces the custom Help menu entry with this variable's value.
FV_USER_HELP_URL	Use to access a custom web page from the FieldView Help menu. The variable's value needs to be the URL as a text string. Only the first 33 characters of this URL will be displayed in the menu entry.
FV_VOLUME_PROBE	Use this to override the default surface-based probe and return probe results based on volume interpolation at surfaces.

(1) Only applies to **FieldView**

(2) Only applies to **FieldView Express**

Reader specific environment variables (e.g., **FV_ACUSOLVE_GRAD**) are documented in [Chapter 1](#) of the **Reference Manual**.

Possible Problems

Why do I get a “command not found” when I try to start FieldView?

Getting a “command not found” error indicates that the directory containing the **FieldView** executable (**FV_HOME/bin**) is not in your command search path. To correct this, either this path should be put in your search path (as described in the **Installation Guide**) or you will need to type the full pathname to start up **FieldView**. For example, if **FieldView** is loaded in the directory **/usr1**, you will need to type the following to start up **FieldView**: **/usr1/fv/bin/fv**.

Why do I get the message “fvbin not found” when I try to start FieldView?

When **FieldView** is installed, the shell script **fv** (located in the **FV_HOME/bin** subdirectory) is created with an environment variable: **FV_HOME**. There are a number of other files which use the explicit value of **FV_HOME** (the installation path), including python files, so the **FieldView** installation should never be moved. Instead, reinstall **FieldView**.

Why will the FieldView process not quit when I type <Ctrl+C>?

The **FieldView** process will not terminate if it was ever placed in the background during the current session.

Balance Between Memory Usage and Performance

When **FieldView** reads data, it performs grid processing after reading the grid. Grid processing increases **FieldView**'s speed in such operations as creating and sweeping Coordinate Surfaces, computing accurate streamlines, and Dataset Sampling. The costs of grid processing are read time and memory.

Below is a list of operations for which performance will be impacted by grid processing:

1. Point probe
2. Dataset Sampling
3. Calculating (and changing scalars/vectors) for streamline & streaklines
4. Setting uniform vectors on Coordinate surfaces
5. Uniformly sampled 2D line plots
6. Changing scalars on imported particle paths
7. Changing scalars on surface flows & vortex cores
8. XDB export of streaklines or particle paths
9. XDB export of surface flows & vortex cores
10. Seed a surface for streamlines (IJK)
11. FVX streamline seeding

Control over the balance between memory usage and probe performance is accomplished with the Grid Processing section of the Data Input panels.

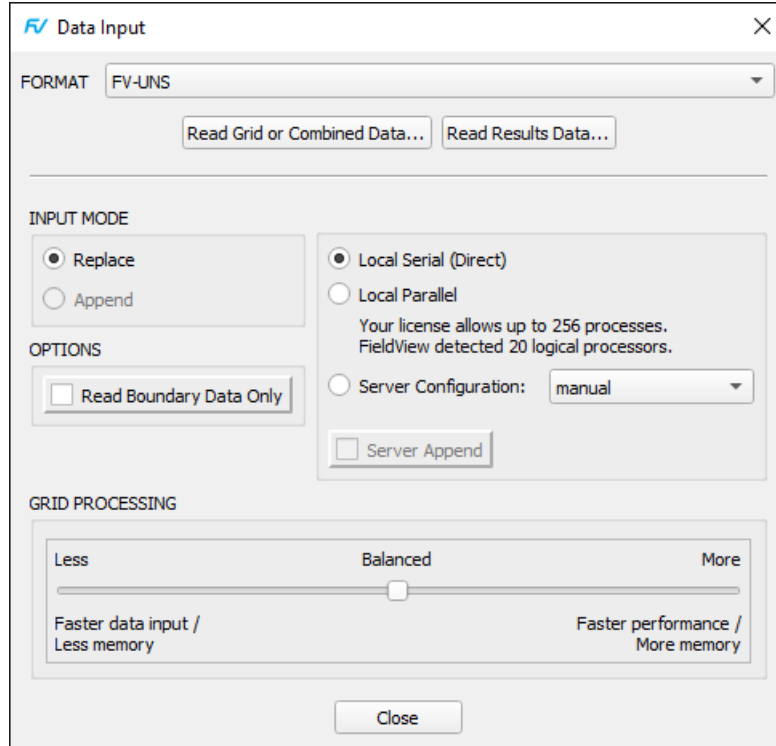


Figure 5 FV-UNS Panel with GRID PROCESSING

Grid Processing setting guidelines:



- If you don't know what setting to use, try "Balanced" (the default value)
- If your data takes a long time to read or if you're going to read many time steps, try "Less"
- If some operations are too slow (see the list of operations impacted by grid processing on [page 25](#)), try "More"
- The recommended Grid Processing option for performing Surface Sampling is "Less"

Alternatively, the environment variables below are still supported:

FV_PROBE_PERFORMANCE

FV_PROBE_SAVE_MEM

The environment variable FV_PROBE_PERFORMANCE can be set to an integer value ranging from 1 to 6.

The correspondence between the Data Input panel settings and the environment variable is as follows:

- Less uses 1
- Balanced uses 3, unless **FieldView** finds that Grid Processing is taking too much memory compared to the unstructured grid size. In that case, a setting half way between 2 and 3 is used.
- More uses a setting half way between 4 and 5.

`FV_PROBE_SAVE_MEM` is still supported for legacy reasons, but it is only recommended for systems with very little memory as it costs a lot in performance. Setting this will force `GRID PROCESSING` to "less", the equivalent of setting environment variable `FV_PROBE_PERFORMANCE=1`, and the Data Input panel will show: Grid Processing controls disabled because `FV_PROBE_SAVE_MEM` is set. while the terminal will now print:

```
FV_PROBE_SAVE_MEM is set. Forcing FV_PROBE_PERFORMANCE to 1.
```

Running Client-Server Mode

FieldView allows multi-process operation in addition to the “standard” (single process) method of using **FieldView**. Multi-process operation permits **FieldView** to split functions that are data- and compute-intensive from those that use graphics and provide control (interactive or script-based). The data- and compute-intensive functions are handled by a program called the “server” and the interactive/control operations are handled by the “client”. Client-server operations use a separate process for the client and each of the servers. This method of operating allows the optimum use of computing resources that are distributed throughout your organization or the world.

FieldView's client-server architecture is unique in that there is a choice as to whether or not to operate in client-server mode or as a single process. In single process operation, the data is read directly into **FieldView**: this mode of operation is called “**Direct**”. **Direct** mode is selected by default.

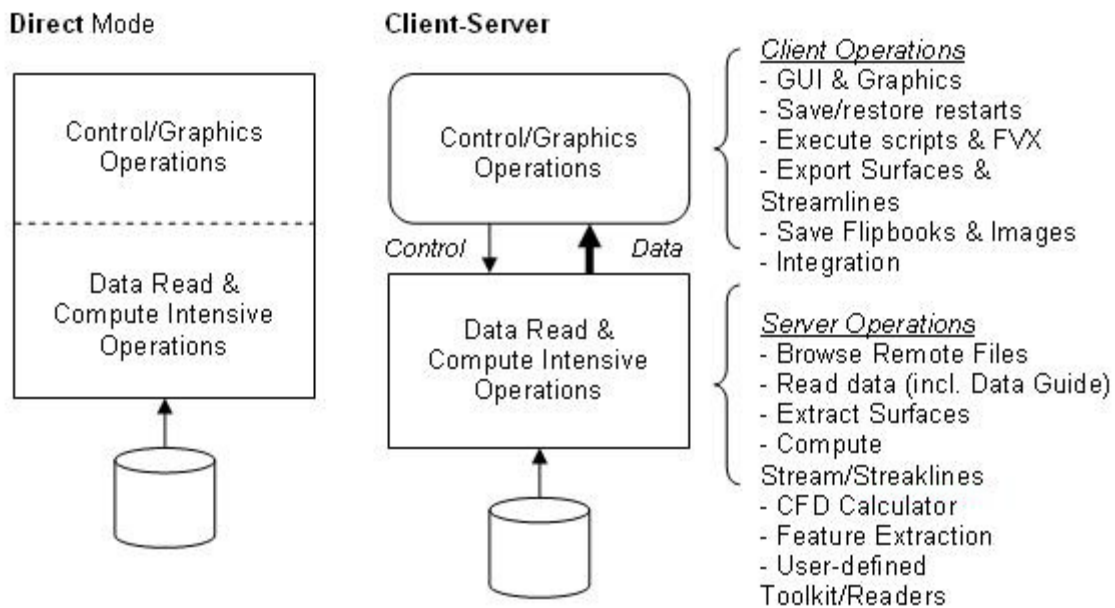
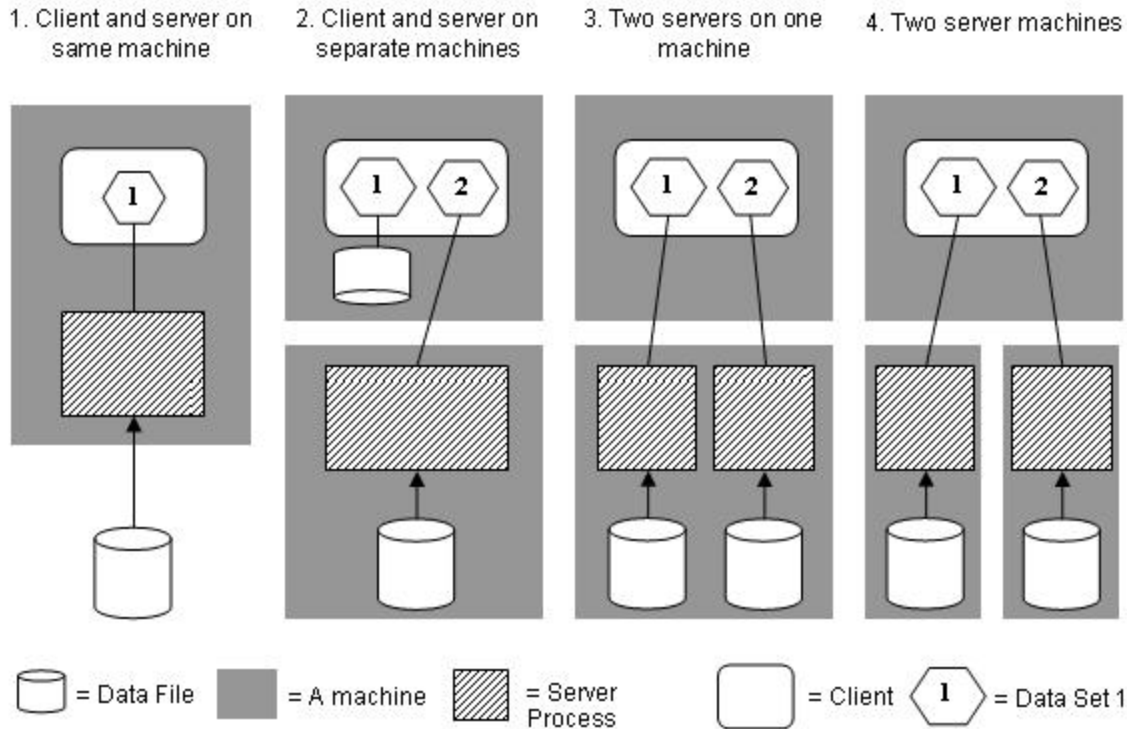


Figure 6 Direct versus Client-Server Mode

When client-server operation is desired, the user simply chooses a server. Each dataset that is read requires a server process. In most cases, **FieldView** will automatically start the server process on the specified machine and read-in the data. This is called “automatic server start”. Other than choosing a server, there are virtually no differences in using **FieldView** in Direct or Client-server mode. When a dataset is replaced or **FieldView** is exited, the server is automatically shut down.

**Figure 7 Client-Server Configurations**

Configuration	Operation	Benefits
Any Client-Server Operation	“Remote” server on local (client) machine or remote machine	A. Gain a separate address space for client and server, so that larger datasets can be read B. Separate address space for each appended dataset, so that many large datasets can be read
1. Client and server(s) on same machine	Select the default “Remote” server, (which is actually the client machine). The server process is started automatically during data read.	See above.

2. Client and server on separate machines, access to a remote and a local dataset in one session	<p>“Local” datasets are read from the client machine (or any file system visible to the client machine). “Remote” dataset is appended by first selecting the “Remote” server machine and then using the remote file browser to locate the file to be read.</p>	<p>C. Remote server access does not require that the file system of the server machine be visible to the client.</p> <p>D. Data files do not need to be copied from a remote server to the local client.</p> <p>E. Large datasets consume memory on the server machine, not the client (desktop).</p> <p>F. Side by side comparison of data that is stored at different physical locations.</p>
3. Two (or more) remote servers on server machine.	<p>“Remote” datasets are read in either replace or append mode by first selecting the “Remote” server machine and then using the remote file browser to locate the file to be read.</p>	<p>A-F above.</p>
4. Two server machines	<p>Same as (3) above</p>	<p>A-F above.</p> <p>G. Each dataset gets its own CPU and memory, thereby increasing performance.</p>

Server startup and control

FieldView is designed to make it as easy as possible to use client-server mode in day-to-day operations. Complete coverage of how to create a server config file is covered in Installing **FieldView** Servers (Optional) in the **Installation Guide**. There are a few key concepts to the architecture:

Server Configuration files	These are simple text files that contain information that FieldView needs to start a server, to set the working directory on the server and to provide explicit control over technical details such as “timeouts”. At least one configuration file must exist for each server. Multiple configuration files can be used for each server to make it convenient to set the working directory and other parameters on a project-by-project basis. The configuration files are read when FieldView is started. See Installing FieldView Servers (Optional) of the Installation Guide for information on Server Configuration files.
Communication via “ports”	The client process communicates to the server process over a mechanism known as ports, which are supported on UNIX, LINUX and Windows. FieldView requires only one port per client process. It is important that the client and server are configured to communicate over the same port. Ports can be specified on the command line, or via environment variable. When the client and server are on physically separate machines, “sockets” are used to “transport” the data between client and server. When the client and server are on the same machine, a more efficient method is used for data transport (“local pipes”).
Automatic Server Start	Once the server configuration files are set up, using Automatic server start is as simple as selecting a server (via a configuration file), selecting the data file type (PLOT3D, etc.) and then using a browser to pick the file to read. It is at this point that a server process is started on the server machine. Whenever the dataset is deleted (either by Replacing it with another dataset or exiting FieldView), the server is automatically stopped. You can use automatic server start when the account that you are logged into is also valid on the server machine. FieldView uses a remote shell command (rsh, remsh, ssh, etc.) to log into the server machine and start the FieldView server.

Manual Server Start	If FieldView is run by default, it will automatically look at the .srv files in the sconfig directory for information about the server. If you instead need to start the server manually, you can use the Manual server that is listed in the File/Data Input/Choose Server menu. When you choose this file, the Manual Start Command Window will pop up listing the server directory, IP address, and port number. Before you hit OK , the server must be running. To start the server, log on the server machine from the client and start the server executable (such as using the fvsrv command). Once the server is running, you can hit OK and FieldView will allow you to read in your data.
One server per dataset	In order to provide maximum flexibility and control, FieldView uses one server process per dataset. One process is used, whether the dataset is transient or steady state. Note that there is no preset limit to the number of servers that can be used simultaneously.
Timeouts	A timeout is an error that occurs when the client (or server) is waiting for an event to take place, and it does not occur within the allotted time. The length of the various timeouts can be set in the server configuration file. The ReadWrite-TimeOut default is 0

Port Use

FieldView will attempt to acquire a port during startup for use in communication with FV servers. This is always done, whether or not you intend to run client-server. By default, it attempts to acquire port number 12345. If it cannot acquire this port, FV will shut down. This can happen when another program is already using this port number, such as another copy of FV running on the same machine or occasionally if **FieldView** is killed rather than exited cleanly. If **FieldView** does not exit cleanly, the port is often tied up for 2-4 minutes. The following is a typical symptom of port unavailability:

```
[Client]: The specified address is already in use.
```

To avoid port conflicts, you can request a different port number by using the **-port** switch on the FV command line. You should keep in mind that ports with numbers below 1000 are reserved. And, the policy for port use is sometimes part of a security protocol. You should consult with your system administrator if you have questions about port usage at your site. Here are examples showing how to specify a single port, or a preferred range of ports where FV will use the first open port in that range:

```
fv -port 21000
fv -port 21000:22000
```

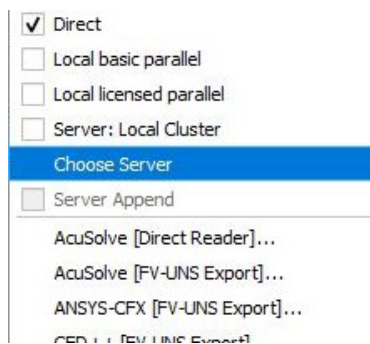
If you will not be using client/server, or the “Local Parallel” options, you can tell FV not to request any port as follows:

```
fv -port none
```

The port number can also be changed by setting the environment variable `FV_DEFAULT_PORT`. If both the environment variable and the `-port` switch are used, the `-port` switch will take precedence.

Data Reading via Client-server

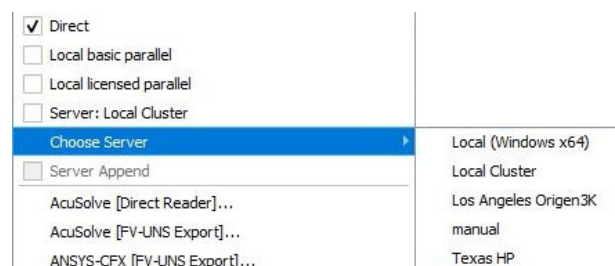
The data input pull down provides control over Direct and Client-server operation, as shown below.



When **FieldView** is first started, the Direct box is checked off. This is because Direct mode is the default way to read data. By the time you reach this pull down, **FieldView** has already read the server configuration files.

To read data without using client-server, you simply select the data file type and you can proceed to browse for the file(s) that you wish to read.

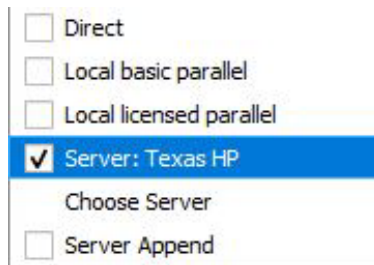
If you wish to use a server to read your data, click on “Choose a server...”, which will let you pick from your selection of server configuration files.



Server configuration files define the server machine, whether automatic or manual startup will be used and the working directory in which you wish to browse for files.

To continue the read-in process, select the server that you wish to use. As an example, we will select “Texas HP”. The server pop-up and the data read pull down will disappear after the selection is made.

At this point, we have selected our server machine and can proceed. Note that you can read a file in one of four ways: via a browser, a datafile restart, a script, or **FVX**. The datafile restarts, scripts, and **FVX** programs are all read from the local (client) file system. Data files are read via a browser that starts on the server machine.



Once a server has been chosen, the Data Files pull down changes as shown to the left. Here, a new line is present, which shows the server “Texas HP” as the current server. At any given time, only one server can be current. Until you change the server (or switch to Direct, by pressing the Direct button), all data will be read via the indicated server.

At this point, you can select the type of file that you wish to read and, at the appropriate time, a file browser will appear, set to the startup path for the selected server configuration files.

At this point, the behavior of **FieldView** depends on the startup mode that has been specified in the server configuration file. This is either Automatic or Manual. Since Automatic startup is *truly* automatic, there is nothing else to say!

Manual Server Startup

Since it is more difficult than automatic startup, Manual Server Startup is used for specific purposes:

- Accessing servers for which you have not set up key pairs to establish passwordless authentication
- Debugging user-defined readers

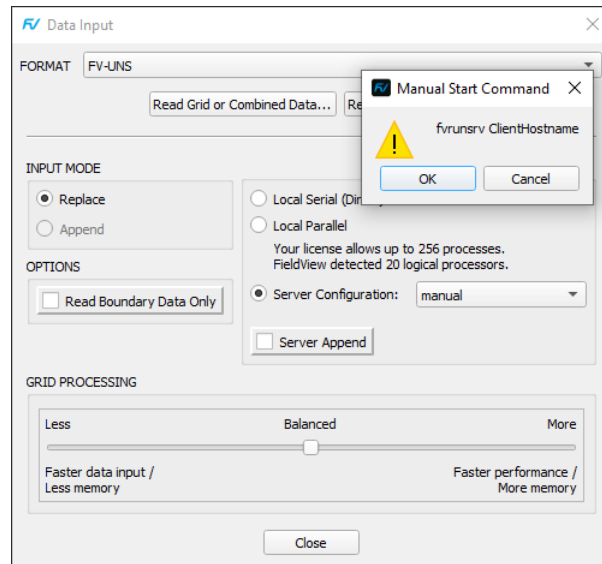
In manual server startup, you will log in to the server machine, and when prompted by **FieldView**, you will start the server process on the server machine. The server process will then communicate to the client via a port.

Manual server start uses a server configuration file which specifies that the server process is to be started manually. Manual start is selected by setting **Autostart: false** in the configuration file for the server machine that you are interested in.

The first step in using manual server start is to log in to the server machine from the client. This is typically done with secure remote shell program `ssh`, or `plink` (Putty.). You begin reading data in the same way as with an automatic start, by selecting a server. When **FieldView** recognizes that the server is to be started manually, you will be prompted to start the manual server. At this point, you log into the server machine, start the server process, and respond “OK” to the server start pop-up. You can then select the data file type and use the browser to pick the file that is to be read. In order to make it as easy as possible to use manual server start, **FieldView** is designed to re-use a server that has been started manually whenever you Replace the dataset on that server. A side effect of this is that you cannot re-use the server that you started manually to Append datasets.

The steps for running a Manual server start are:

1. Start the **FieldView Client** on your Desktop, where for purposes of illustration your Desktop is `ClientHostname`.
2. Log in to the server machine, and change to the directory where you have the FieldView Servers installed. (Note that FieldView Servers are installed along with the FieldView Client, but are also available as “server only” packages.)
3. Use *File.. Data Input..* and on the Data Input panel of your choice, select *Server Configuration: manual*. (Note that reading FVX scripts or restarts specifying a *manual* server will also trigger manual client-server operation.) Then **PAUSE**, when your are prompted with the Manual Server Command prompt, shown below.



4. Start the server by running the **fvrnsrv** program on the server machine, making a connection to your FieldView Client, eg. `ClientHostname`. For example:

```
fvrnsrv ClientHostname
```

The following will be printed in the terminal where you are running the server(s), along with information about plugins loading, and parallel servers starting, if requested:

```
FieldView server Build Date: 01/08/21
Linux x86_64
FieldView server started on servername (192.168.n.n)
```

Note: Command line options are printed for *fvrnsrv* when run without arguments:

```
usage: ./fvrnsrv [-np N] [-hosts host1:N1,host2:N2,...]
        client_hostname [-port NN]
```

Note: Additional command line options appear in [The fvrunsrv Wrapper Program](#) section of the **Installation Guide**. For instance, you can use the `-np 9` command line option to start parallel servers using 8 workers (and one controller server.)

5. Respond OK to the prompt from **FieldView** that is requesting that you start the server
6. Then, the data read with your Data Input file selector will be read with that server.

Please note the following:

- In secured environments, you may need to log in to the server using port-forwarding (eg. the `ssh -R` option.) This is the equivalent to using FieldView's Server Configuration (.srv) `PortForwarding: true` option, as explained in the section *Simple Port Forwarding* of the Installation Guide.
- If you are using a non-default FieldView port (with Client option `-port <port>`, or the environment variable `FV_DEFAULT_PORT`),..the prompt will include `-port <port>`.
- Restarts saved when running manual server will behave the same way when read. (You will need to set up the process manually and use the Manual Start Command dialog.)

FieldView Parallelization with Multithreading and MPI

The hybridized approach to parallelization in FieldView incorporates the use of both multithreading, and MPI driven Parallel Servers. FieldView will take advantage of multithreaded system architectures by default while providing optional controls for thread usage limitations in order that FieldView runs harmoniously with other applications. FieldView Parallel Servers can be used in conjunction with multithreading, with full controls to allow the best post-processing optimization for the target CFD case characteristics.

Multithreading

FieldView will take advantage of multithreaded system architectures. Multithreading is automatic, is the new default, and has been implemented for all modes - Direct, serial and parallel, local and remote client-server.

For Direct Mode or serial Client-Server, when FieldView uses a single process in the MPI sense, the following speed up ranges have been observed thanks to the multithread optimization in the computation of **Streamlines** as compared to earlier FieldView versions, and this is now supported for **all cell types**.

- **Up to 3x** for 4 threads
- **Up to 7x** for 8 threads
- **Up to 13x** for 24 threads

For hybrid parallelization, when both MPI parallel and multithreading have been used, the observed speed up is **up to 3x** compared to earlier FieldView versions using 8 MPI parallel processes each using 6 threads.

Compute of **Vortex Cores, Surface Flows, Separation and Re-attachment Lines** are now also multithreaded, showing the following performance improvements:

- No-slip Surface Flow - **up to 8 times faster!**
- Euler or Offset Surface Flow - **up to 15 times faster!**
- Vortex Cores - **up to 1.8 times faster!**
- Separation Lines - **up to 8 times faster!**

For **Formulas**, FieldView uses a combination of parallelization based on grids within a dataset and nodes within a grid, depending on the operator. The more compute intensive the operator, the bigger the speed up in FieldView, thanks to multithreading. The following performance increases are expected:

- Lambda2 - **up to 14 times faster!**
- Q-crit - **up to 9 times faster!**
- Grad - **up to 6.2 times faster!**
- Less CPU intensive operators (Curl, Div, etc.) - **up to 9 times faster!**

FieldView will use all logical threads by default. Note: for processors on which hyper-threading is enabled, the number of logical threads will be twice the number of physical threads.

We have provided an environment variable `FV_NUM_THREADS` to supplement the more general OpenMP environment variable `OMP_NUM_THREADS`. FieldView will be limited by the lesser of these two settings in the number of threads which will be used. These are useful when you have other processes running on your system (a CFD solver, a pre-processor) and you want to avoid having FieldView compete with them.

Note: Environments variables `FV_ST_STREAMLINES`, `FV_ST_FORMULA`, and `FV_ST_VCORE` have also been provided to *turn off* multithreading for the corresponding features.

Running FieldView with MPI Parallel Servers

Using Parallel Servers in **FieldView** is as simple as using the Client-Server feature. If you have some familiarity with how Client-Server works, then setting up and running **FieldView** Parallel should be straightforward extension. In addition, **FieldView** can run local Parallel depending on whether you have multiple cores and/or a **FieldView** parallel license. Reading data using local **FieldView** Parallel has been described in detail in the section called [FieldView Parallel Datasets](#) in Chapter 1 of the **Reference Manual**.

Similar to the Client-Server feature, you will need to install SERVER programs on to the controller nodes of the system(s) where you wish to run **FieldView** Parallel. (Note that the FieldView Servers are installed along with the *Client*, or with smaller *Server Only* installation packages.) We support both shared memory and cluster configurations.

Again, similar to running Client-Server, you will need to configure files which will permit you to run in the **FieldView** Parallel Server programs. A simple example of a **FieldView** Parallel Server Configuration (.srv) file to run the shared memory mpi version of **FieldView** Parallel on a shared memory system is given below:

```
AutoStart: true
ServerType: shmem
ServerName: my_shmem_system
NumProcs: 9
StartDirectory: /usr2/data/test_PAR
```

This example sets the AutoStart parameter to launch the **FieldView** Parallel server program(s) automatically. As with Client-Server, you can also configure this to run manually. If you do, you will need to execute the `fvrunsrv` command manually.

Note that FieldView also supports the usage of Job Schedulers (eg. PBS) on HPC systems when adding the field `Scheduler:` to your Server Configuration (.srv) file. For more on this subject, please refer to [See “Testing & Configuring FieldView Servers” on page 13.](#) of the Installation Guide.

A total of nine (9) processes has been chosen. This has been set with the command line argument `-np 9` for the `fvrunsrv` command. This means that there will be one con-

troller process (which acts solely as a dispatcher) and eight (8) worker processes (which read data) started on this shared memory system named ***my_shmem_system***. The `fvrunsrv` command also starts the **FieldView** Parallel server program, `fvsrv_par`.

A start directory, `/usr2/data/test_PAR`, is set so that when the file browser to read data comes up, it will be started at this location.

A simple example of a **FieldView** Parallel Server Configuration (.srv) file to run **FieldView** Parallel using a system such as a LINUX cluster might look like:

```
AutoStart: true
ServerType: cluster
ServerName: my_cluster_system
NumProcs: 5
StartDirectory: /usr2/data/test_PAR
MachineFile: /path/to/machines.txt
```

In this case, again the `AutoStart` option has been chosen.

For a parallel cluster, we will need to have the MPI and **FieldView** Parallel Server programs installed on the controller node of the system, in this case called ***my_cluster_system***. The installation location for this set of files can be anywhere you choose. When this **FieldView** Parallel Server is selected, a total of five (5) processes will be run. Again, one of these will be a controller process, and the remaining four (4) will be used to read and process the dataset.

For the cluster option, MPI will use the default “machine” file `openmpi-default-hostfile`, found within the MPI installation with **FieldView**. If you do not want to let MPI use defaults and would rather specify which nodes of your cluster will be used as **FieldView** “worker” servers, you can:

For Server Configuration (.srv) files, use the `MachineFile:` field in a Server Configuration (.srv) file to specify a custom machine file in the Server Configuration (.srv) file.

Note: For Manual client-server, specify your parallel “worker” nodes using the `-machinefile` or `-hosts` argument to the command **fvrunsrv**.

Note also that all nodes of a cluster must be able to resolve the path used for the field `ServerDirectory:` in the Server Configuration file (.srv) in order to find the program `fvsrv_par`.

Please [See “Testing & Configuring FieldView Servers” on page 13](#) of the Installation Guide for additional details if needed.

To read data, go to the Data Input pulldown menu and use *Server Configuration* to select your Server Configuration (.srv) file.

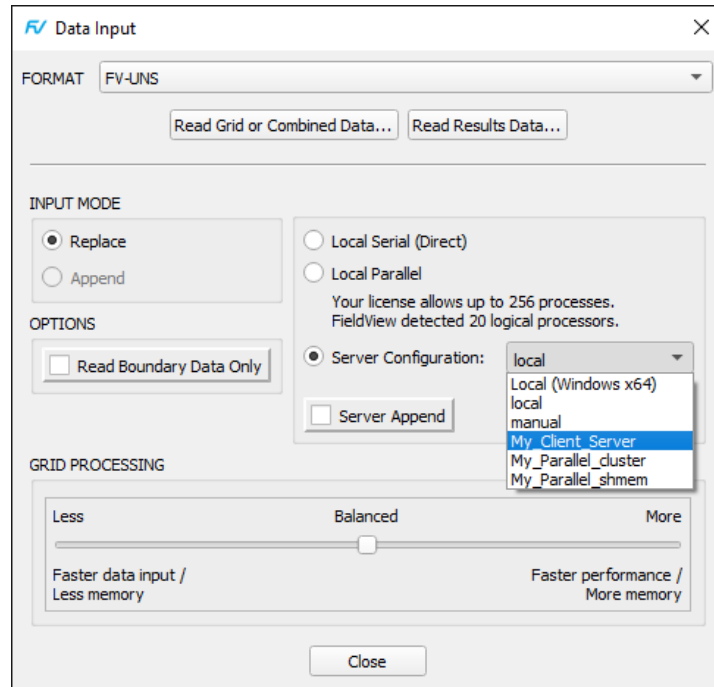


Figure 8 Server Selection from the Data Input Menu

If both examples for the server configuration files are placed in the `/fv/sconfig` directory, then when the Choose Server option is selected from the Data Input menu, both the cluster and shared memory based **FieldView** Parallel options to read data will be present (as illustrated above).

After this point, we can now read the data via a browser which starts on the Controller server system.

Q. How can I tell if my dataset has multiple grids?

If you want to quickly determine whether a dataset has multiple grids there are a few simple observations that can be made by attempting to read the data in the Direct Mode of operation (not Parallel or Client-Server). First, if a PLOT3D file has multiple grids specified, and the read is successful, then you can determine how many grids are available by reviewing the Grid Subset Selection Panel. Also, once the

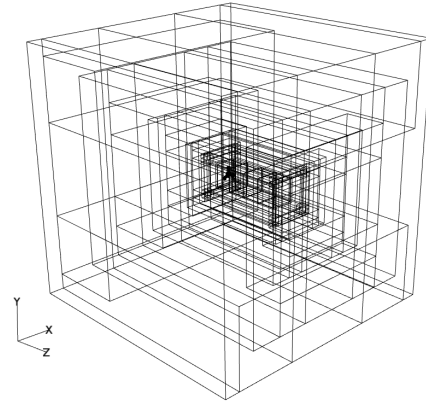
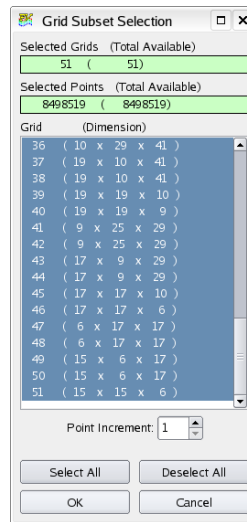


Figure 9 Grid Subset Selection and Outline

dataset has been read in successfully, you can see the outline of each of the individual grids in the graphics window. For the case of unstructured data, the number of grids for a given dataset, as well as the number of nodes and elements for each grid will be listed in the console window. There will be one line in the console window corresponding to each grid. A typical console window output might contain the following lines:

```
Unstructured grid 1 has 81830 nodes and 12614 elements.
Unstructured grid 2 has 68350 nodes and 12191 elements.
Unstructured grid 3 has 61992 nodes and 12449 elements.
Unstructured grid 4 has 55450 nodes and 11502 elements.
Unstructured grid 5 has 79576 nodes and 12545 elements.
Unstructured grid 6 has 81502 nodes and 11656 elements.
Unstructured grid 7 has 82392 nodes and 12786 elements.
Unstructured grid 8 has 74937 nodes and 12813 elements.
Unstructured grid 9 has 51221 nodes and 9562 elements.
Unstructured grid 10 has 54713 nodes and 9574 elements.
Unstructured grid 11 has 49797 nodes and 9172 elements.
Unstructured grid 12 has 45543 nodes and 8938 elements.
Unstructured grid 13 has 48358 nodes and 9769 elements.
Unstructured grid 14 has 48633 nodes and 9696 elements.
Unstructured grid 15 has 52293 nodes and 9633 elements.
Unstructured grid 16 has 43283 nodes and 8838 elements.
```


*Q. Can **FieldView** create a multigrid dataset from a single grid dataset, or repartition existing grids?*

Yes, FieldView's Auto Partitioner can create a multigrid dataset or repartition existing multi-grid PLOT3D or OVERFLOW-2 input data. For more details, see the [Auto Partitioner section](#).

*Q. Can **FieldView** create partitioned files from a multigrid dataset?*

At the present time, there is no feature in **FieldView**, or standalone utility capable of creating partitioned files from a multigrid dataset.

FieldView Parallel Performance Results

The degree to which certain operations in **FieldView** Parallel will scale depends upon several factors pertaining to the dataset, and may also depend upon hardware issues as well. As such, it is difficult to make generally applicable statements concerning the performance of **FieldView** Parallel. Some testing results are included here. **FieldView** Parallel testing was carried out on a set of test cases run using FLUENT 6.3.26 and Acusim 1.7e. Boundary results were present in the FLUENT FV-UNS exports. Details for four benchmark cases, all containing 38 grids, and made up entirely of either tetrahedral cells or arbitrary polyhedral cells, are summarized in the table below:

	Tet Benchmark (small)	Tet Benchmark (large)	Arb Poly Benchmark (small)	Arb Poly Benchmark (large)
File Size [bytes]	420,111,352	811,480,068	1,847,041,196	3,555,019,756
Nodes	2,883,166	5,526,084	16,831,972	32,389,866
Elements	15,540,893	30,374,480	2,932,112	5,589,608

Table 1: Summary of **FieldView** Parallel Test cases

Timing tests were performed using an HP xw8600 system with 4 dual core CPU (Intel X5460, 3.166GHz). The system was configured with 32GB of RAM. Elapsed timings were obtained running Parallel Shared Memory (shmem). Averaged results on scale-up factors recorded for each of the benchmarks are presented below.

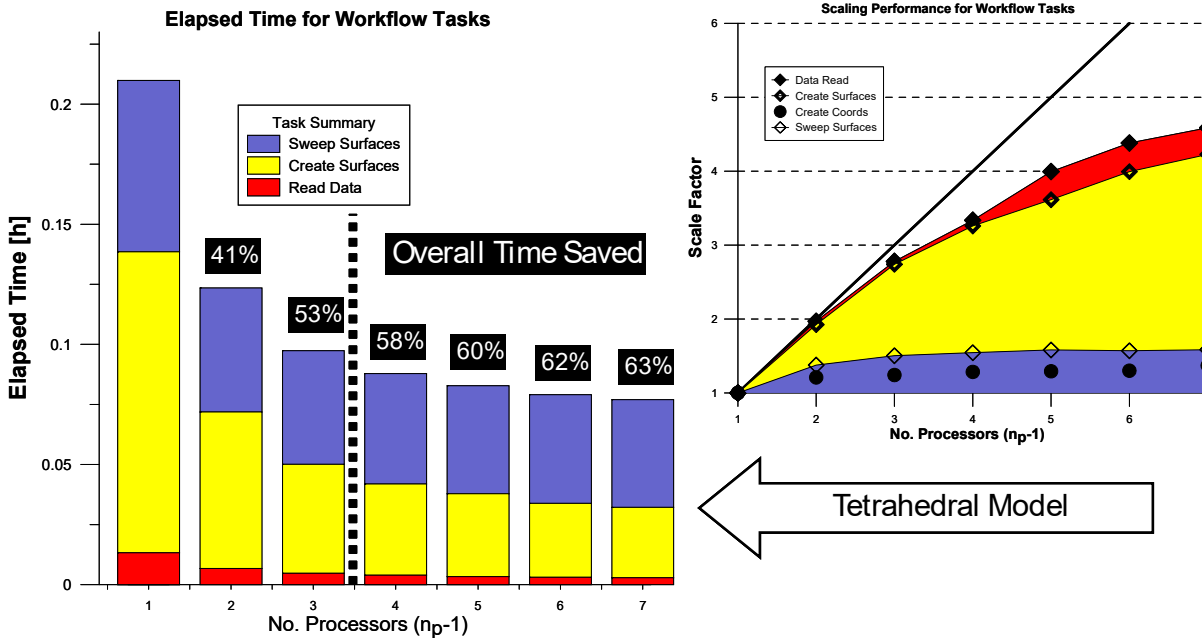


Figure 10 FieldView Parallel Scale-Up for Tetrahedral Data

Benchmark timings were obtained for the tasks of reading data, creating coordinate and

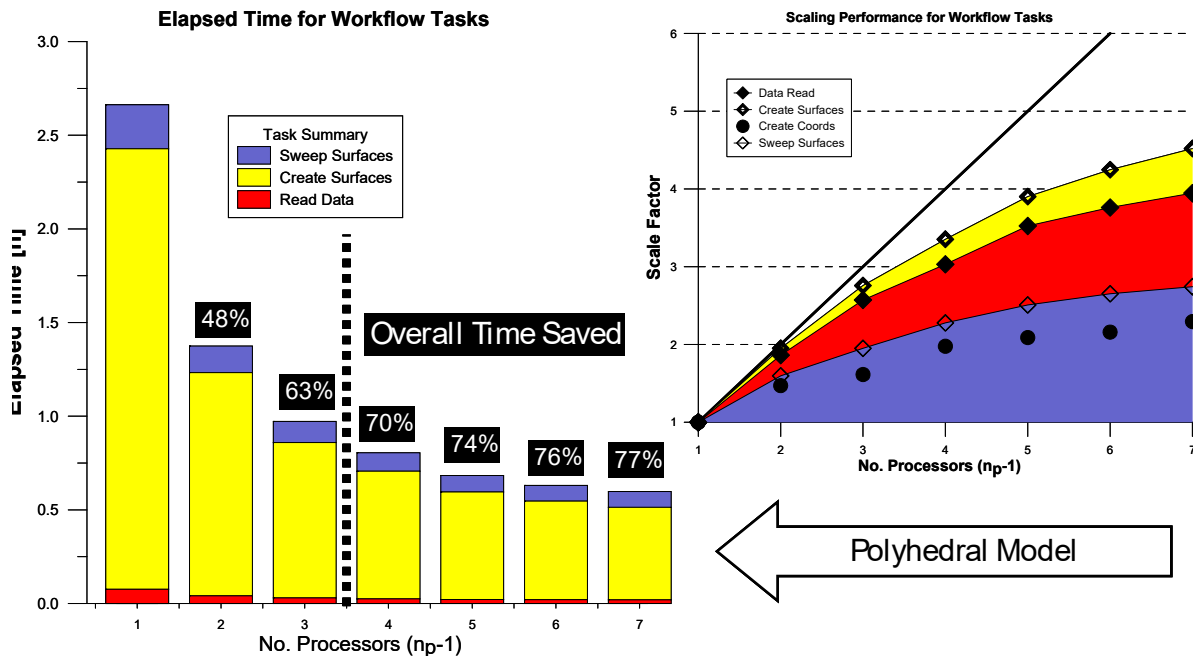


Figure 11 FieldView Parallel Scale-Up for Reading Data

iso-surfaces and sweeping these same surfaces. Overall time savings are reported as a percentage relative to the time required to complete these tasks using **FieldView** in Cli-

ent-Server mode. Using the just *three* worker processes (for np values to the left of the dotted line) we expect to see scaling improvement of 2x or better.

Benchmark timings were also carried out on a partitioned FV Unstructured dataset composed of 53 grid and results files with a total of 153 million nodes. The test system was an HP 740C with 8 blades or compute nodes. Each compute node was configured with an Intel XEON 4 core (Intel X5260, 3.33GHz, dual core, dual processor), a local file system and 16GB RAM. Local file systems were connected with a GigE network switch.

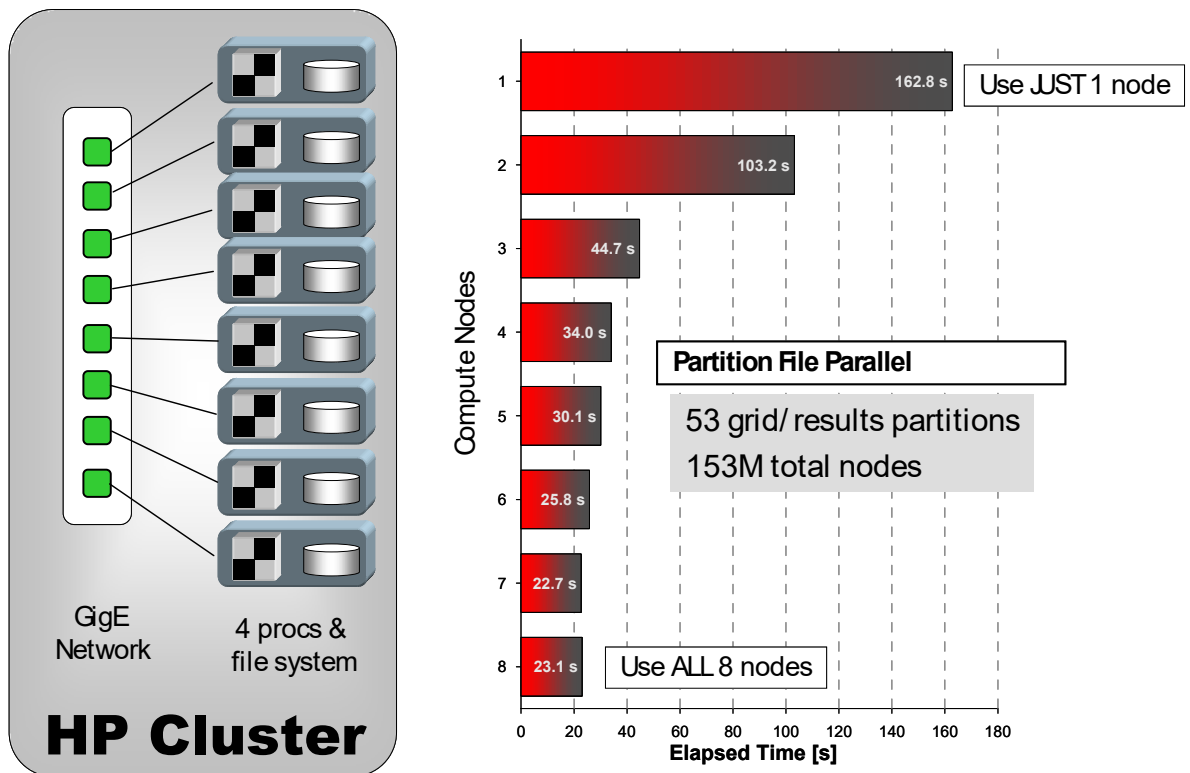


Figure 12 FieldView Parallel Scale-Up for Reading Data

Timing tests were obtained as follows. For the case of one compute node, all 53 partitions were read from the local file system on that node. For two compute nodes, the 53 partitions were spread across the file systems for two nodes, and the dataset was read again. By utilizing all 8 compute nodes, a 7X reduction in the time required to read data was achieved.

Obtaining Timing of Various Postprocessing Operations

Owing to the complexity and diversity of datasets, networks and file systems, it is often difficult to identify where problems may be occurring during postprocessing operations. To understand how effectively **FieldView** (Parallel or Direct) may be running for a given dataset in a given environment, we provide a *Script Command* **TIMING ON/OFF** (which can be nested) to report CPU and Elapsed timings for any (scripted) operation. Below, two examples are shown for standard '.scr' script (Chapter 5, Reference Manual), then FVX '.fvx' script (Chapter 4, Reference Manual.) The second example is suitable for Python '.py' scripts:

```
timing on
restart data data_restart_file.dat
timing off Comment

fv_script("timing on")
fv_script("restart data data_restart_file.dat")
fv_script("timing off Comment")
```

The above examples show how you can add **Comment** to give your output context. The following output example shows how this will be used, and how the timing reports will look:

```
[Client]: Comment: CPU time = 0.05 + 0.00 = 0.05 Elapsed time = 0.35
Page faults = 0 per elapsed sec = 0.00 Swapouts = 0
Context switches: voluntary = 21 involuntary = 0
```

Note that timings are taken at the FieldView Client, thus CPU time may be very small as compared to Elapsed when operations are performed on a remote FieldView Server. Interpretation of the timing information that **FieldView** produces will be involved since the performance of **FieldView** will be dependent not only on the hardware environment but also on the type of datasets being examined and the tests actually being performed. If you encounter what you believe are performance problems with **FieldView**, we recommend that you send your complete timing output and testing methodology to us so that we can analyze the results for you. You can direct all output to a file with most unix shells using the following command line syntax:

```
fv -s my_script.scr 2>&1 | tee my.log
```

Auto Partitioner

FieldView is capable of automatically partitioning your structured datasets imported using the PLOT3D and OVERFLOW readers. This will enable MPI parallel speedups even if your data is single grid or has poorly balanced node counts across grids.

This new and unique feature is available to all customers with licenses for FieldView Parallel 32 and above. If your license does not allow the Auto Partitioner to be enabled and you would like to try it with your specific datasets and hardware environment, please contact sales@tecplot.com.

When a parallel compatible read option has been selected, with either local or remote parallel servers, the Auto Partition option will be enabled on the PLOT3D and OVERFLOW reader panels.

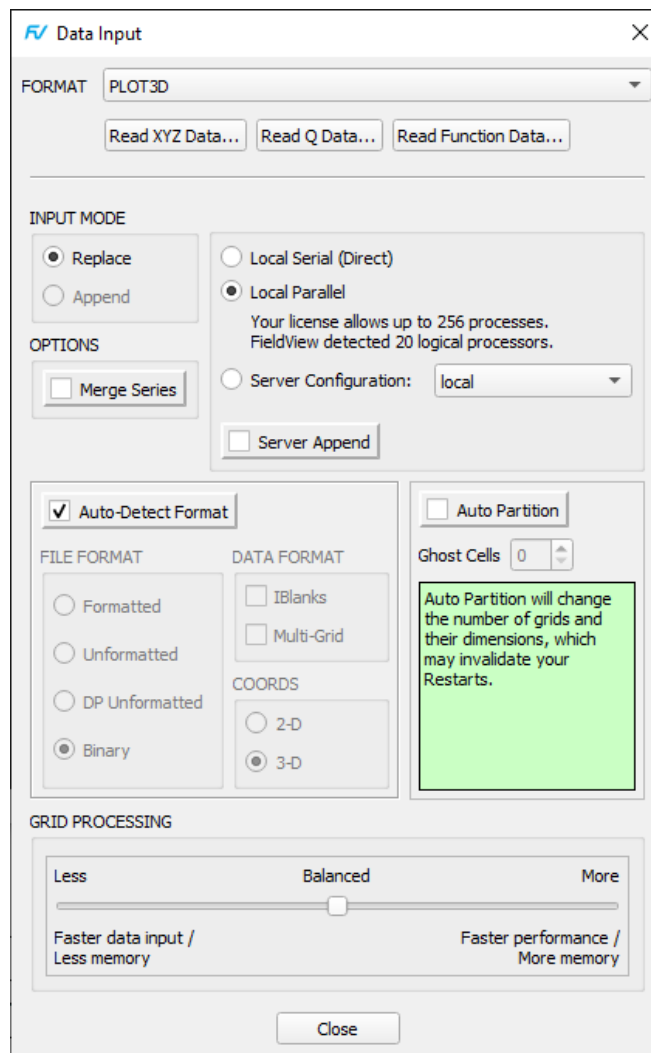


Figure 13 Partitioner enabled but set to OFF.

When Auto Partitioner is set to ON, FieldView will automatically compute an optimal partition for your dataset based on the number of FieldView MPI workers and the number of nodes per grid. The updated grid list will be immediately visible from the Grid Subset Selection panel.

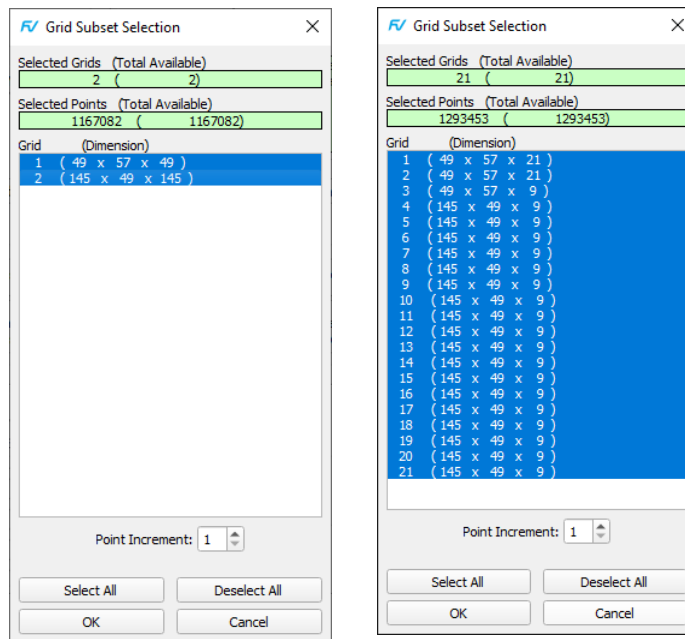


Figure 14 Here, we see a case with its original grid list at left, and at right, the new grid list obtained by the Auto Partitioner.

A report of the partitioning operation will also be printed to the console window, showing which grids have been split and which new grids have been produced.

```
The auto partitioner created 19 new grids
Grid 1 is now grid 1, 2, 3
Grid 2 is now grid 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19,
20, 21
```

If Auto Partition is ON but no further partitioning is needed (grids are already sufficiently well balanced for the number of FieldView MPI workers), the Auto Partition step will be skipped.

Auto Partition adds very little time and memory overhead while significantly improving the time needed to read the dataset and to create most post-processing objects, thanks to improved parallelism.

However, users should be aware of two drawbacks:

- Grids that get split by the Auto Partitioner appear to FieldView as new grids. It means that any operation relying on IJK or grid numbers will be impacted by this feature. For instance, if a Computational Surface Restart has been saved with Auto Partitioner OFF and is then loaded back after a dataset read with Auto Partitioner ON, it may

lead to the generation of different surfaces. Features that don't specifically rely on IJK and grid numbers (a majority of features) will not be affected.

- New partitions between grids which did not exist in the CFD solver will be created. This will cause discontinuities in User Defined Variables computed by FieldView that include spatial derivatives. These discontinuities could be visible when using scalar coloring on surfaces or when plotting Iso Surfaces (see below).

To help with the latter, users will have the option to introduce one or two layers of ghost cells between newly formed partitions. This setting will be applied at read time.

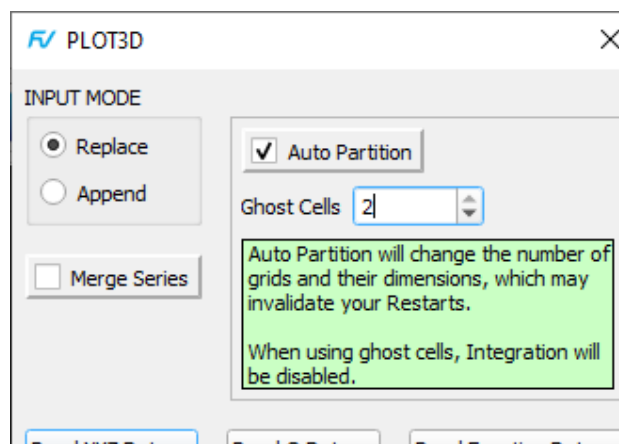


Figure 15 Auto Partition Ghost Cells specified.

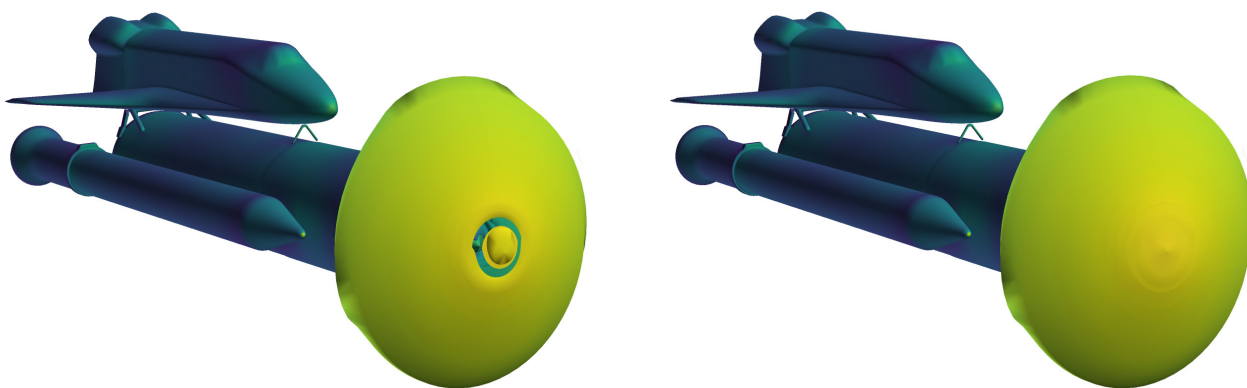


Figure 16 An Iso Surface visualized after partitioning, with image at left showing use of no Ghost Cells, and 2 Ghost Cells at right. See how the gap artificially created by the Auto Partitioner got closed.

Because these layers of ghost cells will be added on both sides of the partition, this setting will actually create cell and face duplicates in the mesh, which could affect quantitative results computed by FieldView. For this reason, surface integration will be disabled when ghost cells are used (as shown by the warning message in Figure 4).

Structured Boundary Files (.fvbnd) are fully supported with Auto Partition, with or without ghost cells.

Auto Partition settings will be saved in the Data Restart (.dat) file. When a dataset is imported back into FieldView with such a Restart, FieldView will automatically adjust the partitioning to the number of MPI workers. This ensures the best performance even when a Restart was saved on a system and then used on another system.

The Data Input table fields *auto_partition* and *ghost_cells* for PLOT3D and OVERFLOW2 provide control of Auto Partitioning in FVX scripting. Please refer to the [Table on page 169](#) of the *Reference Manual*

Other limitations to the Auto Partitioner:

- Only 3D PLOT3D and OVERFLOW data in the Binary, Unformatted and DP Unformatted formats are supported. 2D and Formatted formats are not supported.
- Grids and Results must be read using the same format.
- Face Data and Function files (.fvsrcf) are not supported.
- Grids described by brick files will be left untouched by the Auto Partitioner.

Chapter 2

FieldView

Tutorial

Introduction



The following chapters are designed to give both the new as well as the experienced **FieldView** user a detailed look at some of the abilities of the product. You will be guided through a given dataset and shown the steps necessary to create the image(s) shown on each page of the tutorial. Each tutorial covers a different application or industry. We have tried to create visualizations and perform calculations that might be useful for an individual in that discipline. It is not expected nor required that you work through all of the tutorials to learn **FieldView**. More experienced **FieldView** users may wish to browse through the tutorial chapters and the accompanying images to learn new visualization techniques.

Prerequisites

The tutorials have been written with the assumptions that you have become familiar with the [FieldView Interface](#) and [Basic Operations](#). In general, it will be helpful to understand the following basics:

1. How to find the data readers in the **File** pull-down on the Main Menu and bring up the desired reader panel for data input.
2. How to find the visualization panels either from the **Side Toolbar** or the **Visualization Panel** pull-downs on the Main Menu to create and modify surfaces and rakes in **FieldView**.

3. How to change Scalar, Vector and Threshold functions for surfaces and rakes directly on their respective visualization panels, and/or bring up the Function Specification panel to “load-up” different functions in the function registers (Iso-surface, Scalar, Vector, and Threshold).
4. How to change the Geom (geometry) color and colormaps used to color surfaces on the Scalar Colormap Specification panel.
5. How to move the data around the graphics window using mouse actions to translate, rotate and zoom in to the data.

Tutorial Data

The tutorial data is not installed by default with the **FieldView** software. It resides on the **FieldView** DVD in its own directory called `/tutorial` on the top level of the DVD. All of the files required for each tutorial can be found in separate directories in `/tutorial`, named for each of the applications. The tutorial directories and their approximate sizes (disk space required) are listed alphabetically in the table below. All of the tutorials contain **FVX** scripts (see the section on these scripts below).

Tutorial Name	Space Needed (Mb)	Notes
/aerospace see Basic Aerospace Tutorial	8	Introductory PLOT3D case for external aerodynamics, keyframe example described, FVX example uses this case for lift, drag, moment.
/casting see Free Surface Flow Tutorial	97.2	Introductory FLOW-3D(R) case for open channel flow, highlighting transient data handling
/automotive see Basic Automotive Tutorial	2	Basic flow in a valve, simple case
/turbomachinery see Basic Turbomachinery Tutorial	14.1	Flow in a single blade passage, highlighting periodic streamlines, and numerical integration of mass flow
/combustion see Basic Combustion Tutorial	11.3	Simple combustion case, highlighting streamline calculations, and flame envelope using iso-surfaces.

/vortexshedding see Vortex Shedding Tutorial	52.6	Transient flow around a cylinder, with description of streakline calculation. FVX time history and keyframe illustration accompanies example.
/polymer_processing see Polymer Processing Tutorial	5.7	Flow of a non-newtonian fluid in a Kenics mixer, highlighting extensive use of Function Calculator. FVX blade shear histogram and keyframe illustration accompanies example
/biomedical see Biomedical Tutorial	4.3	Internal flow in a complex manifold device, illustration of how to calculate a species mass balance, how to perform dataset mirroring.
/chemical_processing see Chemical Processing Tutorial	8.3	Flow in a mixing tank with multiple impellers, visualization of flow patterns.
/autodiesel see Automotive Diesel Combustion Tutorial	24	Transient flow in a diesel piston based on a standard KIVA tutorial case, plotting fuel concentration vs crank angle
/feature_extraction see Feature Extraction Tutorial	5	Complex internal/external flow around and inside a droplet. Feature extraction tools are highlighted in this tutorial. A Keyframe illustration accompanies this example.
/heat_exchanger see Heat Exchanger Tutorial	5	Flow inside a heat exchanger for two different process configurations. Handling of multiple datasets, with numerical dataset comparison is highlighted here.
Streamline Seeding Tools see Streamline Seeding Tools	NA	This tutorial uses the above datasets to illustrate the use of the FVX utilities for seeding streamline rakes using spherical, cylindrical or rectangular patterns. (see /util/fvx on the FieldView DVD).
/working_with_xdbs see Working with XDBs Tutorial	41.3	How to create and import an XDB dataset. Use of XDB datasets for Dataset Comparison.

Some of the data files used in the tutorials have been renamed and no longer carry usual extensions. This is to avoid filenames with more than one period "." in them (due to legacy considerations and DVD writing limitations). For example, the CFX-4 data used in the Chemical Processing tutorial no longer has the .dmp extension. When reading in

some files, the data file may *not* initially show up in the file browser due to the use of file filters. The CFX-4 file browser assumes files end with the `.dmp` extension and its file filter reflects this. To see the data file, change the file filter to `"*"` instead of `".dmp"`.



Note: Users are advised that data read-in times are somewhat slower from the DVD as opposed to reading the data after it has been transferred to your local disk.

There is additional sample data that is part of the default **FieldView** installation. This data can be found in the `/examples` directory in the **FieldView** installation area (`/fv/examples`). There is also a demo using the F18 dataset used in [Chapter 3](#) - Basic Aerospace Tutorial which can be found in the `/demo` directory (`/fv/demo`).

Working Through FieldView Tutorials

It is assumed that you have read the chapters mentioned in the Prerequisites section above before you run the tutorials. The tutorials have been written as explicitly as possible without making them excessively long. If a step requires actions that have been performed in previous steps, the instructions may be less explicit than in the first instance.

Each of the Tutorials features a 'What you will learn' section to start off the chapters. The majority of the tutorial steps have been designed so that the instructions, panels and images appear on a single, self-contained page. The tutorials are designed so that the image that appears in the graphics window should be the end result of following *all* of the instructions on the page as well as having performed those steps on previous pages (if not at the beginning of a tutorial). That is, the image at the top of each page of each tutorial (for the most part), is the *end result* of the set of instructions on that page. Also, transformations (rotations, translations, zoomings) are not explicitly shown as panel actions. Instructions are given so that you can obtain a view similar to that shown in the image. Matching the view exactly is neither important nor necessary.



Note: Tutorials are documented with white backgrounds and black text. However, the default background color in **FieldView** is black. We recommend that you change your background color to white as soon as you start on a Tutorial. To change your background bring up the Background Color panel from the View pulldown menu.

We strongly recommend that you save a Complete Restart after each page of a given tutorial so that they can easily re-create the image at a later date. A complete restart is a set of files containing information about the current state of **FieldView**. It saves information about the data file read in (so it will refer to the DVD if the data is read from the DVD directly) and any surfaces/rakes that have been created as well as their attributes. Restarts can be created at any time, of course, but one complete restart for each step will assure its availability later. For information about how to save a Complete Restart and other types of restarts, please refer to [Chapter 5](#) of the **Reference Manual**.

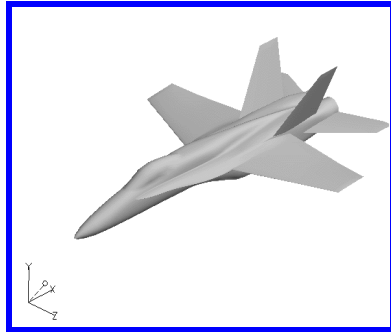
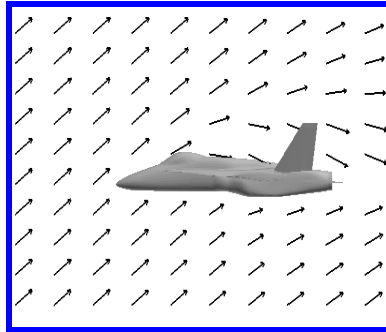
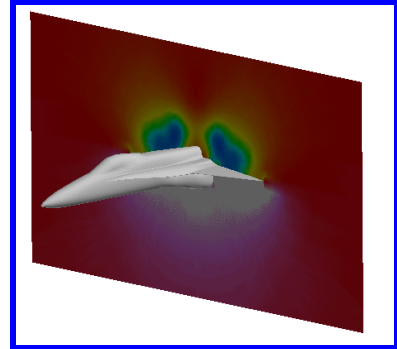
Due to a desire to limit repetitiveness, not all functions and actions are represented by panel images. That is, if a tutorial step has the user create 4 coordinate surfaces, the end result Coordinate Surface panel may be shown (showing 4 surfaces), rather than 4 views of the same panel with successive surfaces being created.



Note: The tutorials were each designed to be run right after initially starting **FieldView**. If other data has been loaded into **FieldView** and visualizations performed, or if other tutorials have been worked before starting a given tutorial (without restarting **FieldView**), then panels may exist in states different than that described in the instructions. If you are just starting with **FieldView**, we recommend restarting the program for each tutorial to avoid confusion.

The following sections may be helpful in guiding you to specific postprocessing tasks which are covered in each tutorial.

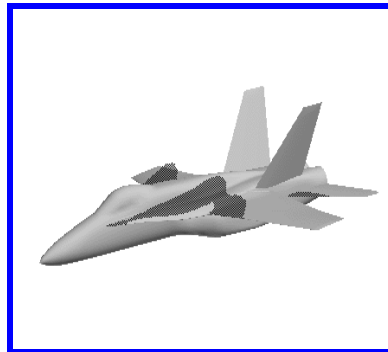
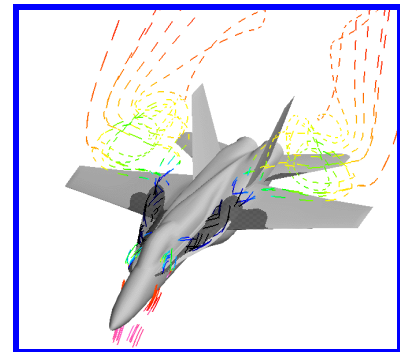
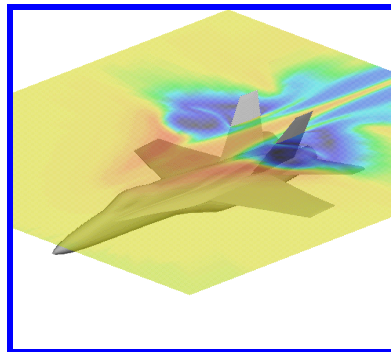
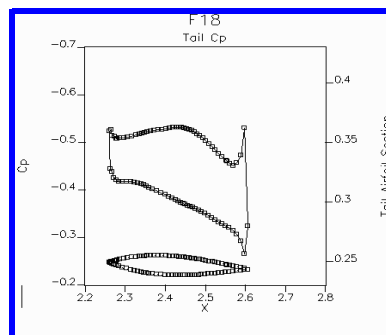
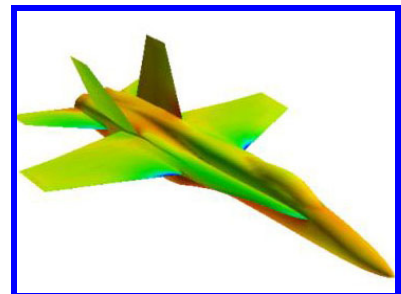
Working with Aerospace Data

**How to create Boundary Surfaces
and How to Mirror a Model****How to display Velocity Vectors
on a Surface****How to create Pressure
Profiles**

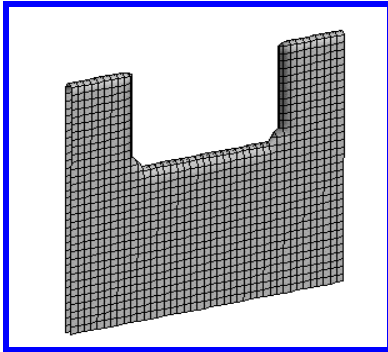
**How to create and use functions
for displaying properties**

How to use the global Color Map
See "Step 4: Display the Pressure
Coefficient" in the User's Guide

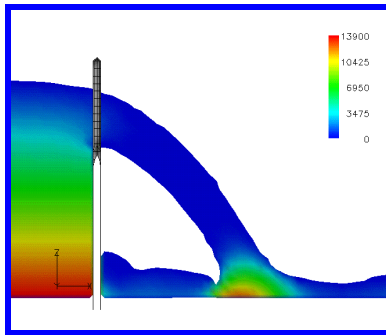
**How to create a Transparent Sur-
face**
See "Step 5: Visualize the Vortex
Core" in the User's Guide

**Creating an Iso-Surface to dis-
play the pressure distribution****How to create Streamlines****How to make a transpar-
ent surface****How to make a 2D Plot****How to create a Keyframe Ani-
mation**

Working with Free Surface Flows



How to display a mesh on a surface



How to read in FLOW3D data into FieldView

How to visualize a transient dataset

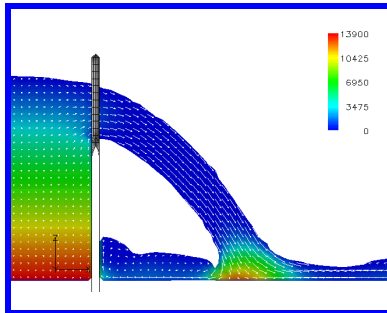
See “Step 2: Visualize Transient Free Surface Flow” in the User's Guide

How to create an IJK surface with the Computational Surface Panel

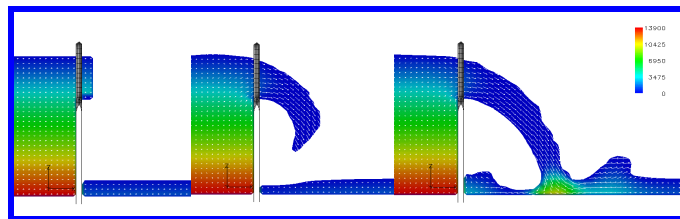
See “Step 2: Visualize Transient Free Surface Flow” in the User's Guide

How to use the legend

See “Step 2: Visualize Transient Free Surface Flow” in the User's Guide



How to display velocity vectors on a scalar surface

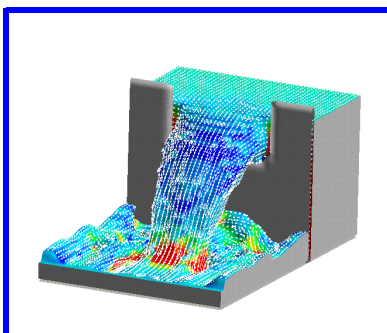


How to create a transient flipbook animation

See “Step 4: Create a Transient Flipbook Animation” in the User's Guide

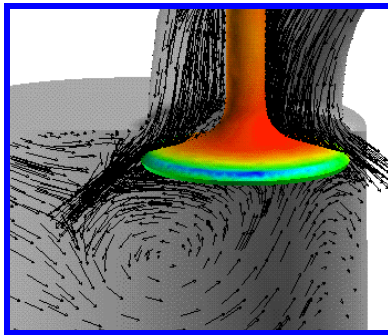
How to clip a surface by using Thresholding

See “Step 2: Visualize Transient Free Surface Flow” in the User's Guide

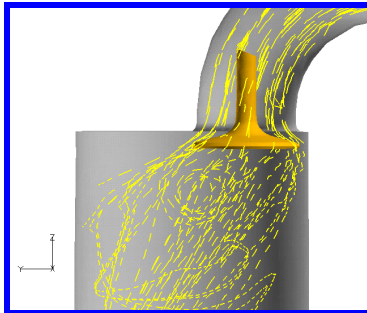


How to automatically open a panel by “Quick-Picking” its surface

Working with Automotive Models

Basic Automotive

How to visualize vortex formation using velocity vectors



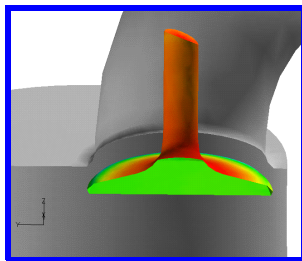
How to create a flipbook animation and save it to a file

How to change velocity vector settings

See "Step 2: Capture Vortex Formation with Velocity Vectors" in the User's Guide

How to cut away a portion of a surface

Step 1: Read in FLUENT-UNS Data



Integral of current boundary surface:

NOTATION

S (scalar) = pressure
Int(S) = surface integral of S
A = total surface area

RESULTS

Int(S) = 361.131

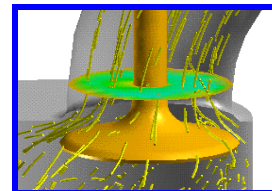
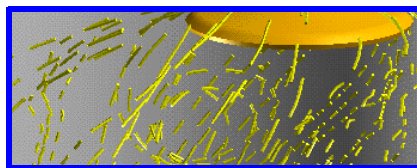
A = 0.00388749

Average of S = Int(S) / A = 92895.7

NOTE: Normals are absent. No normal-based integrals.

How to integrate a scalar on a surface

Also See "Step 6: Integrate Pressure" in the User's Guide

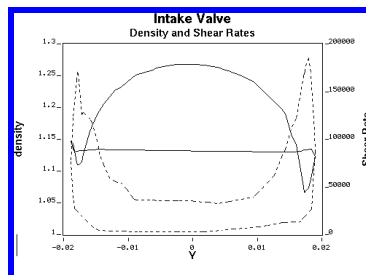
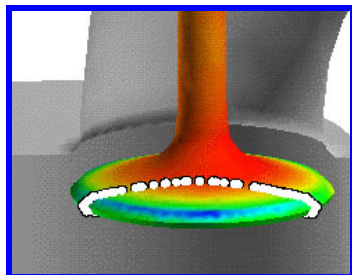
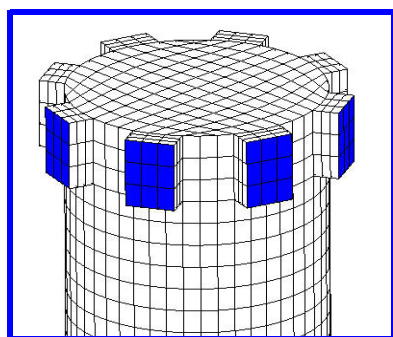


How to visualize vortex formation using streamlines *and*,
How to automatically seed a planer surface *and*,
How to display shaded streamlines

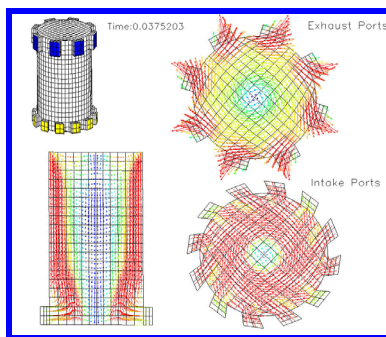
Also See "Step 3: Display Vortex Formation with Streamlines" in the User's

How to create a 2D plot of shear rate and density on an a valve

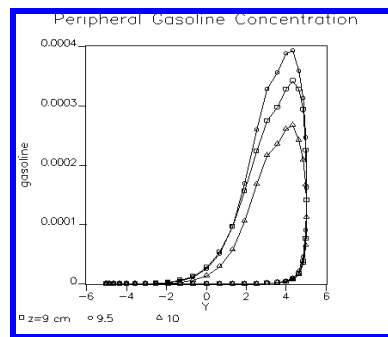
See “Step 5: Create a 2D Plot Along a Surface” in the User's Guide

**Diesel Combustion****How to create a hidden line view of a Boundary Surface**

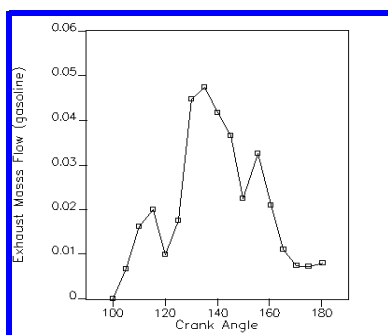
Also See “Step 1: Create a Hidden Line View” in the User's Guide

**Display velocity vectors and the gasoline concentration as a function of time.**

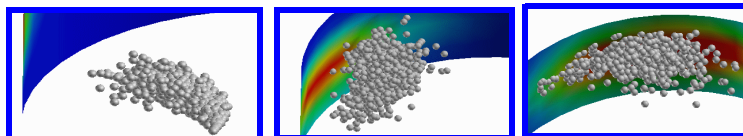
Also See “Step 2: Viewing Velocity Vectors and Fuel Concentration” in the User's Guide

**2D plots of the gasoline and O2 concentrations**

Also See “Step 4: 2D Plots of Gasoline and O2 Concentration” in the User's Guide

**Functions and scripts to display the exhaust mass flow rate**

Also See “Step 5: Display the Exhaust Mass Flow as a Function of Crank Angles” in the User's Guide

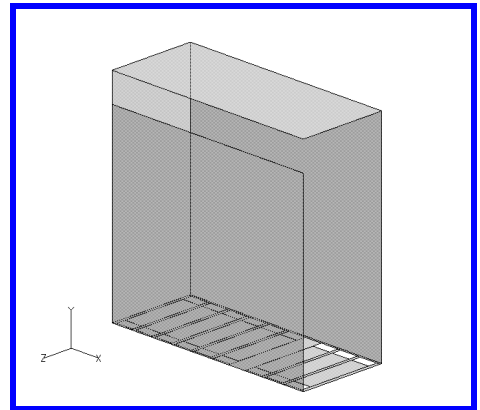
**How to amplify the contour range of gasoline and How to make an animation of the transient spray results**

See “Step 3: Amplifying Contour Range and Animating Spray Results” in the User's Guide

Working with Combustion Models

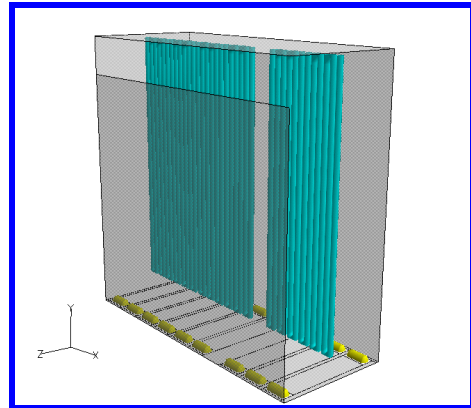
How to create wall boundaries in your PLOT3D data file

See [“Step 1: Create Wall Boundary Files”](#) in the User's Guide



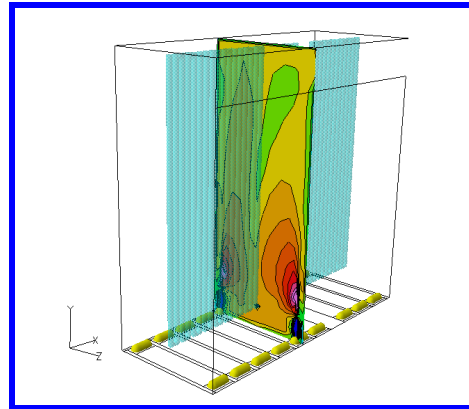
How to create walls using Iso-Surfaces

See [“Step 2: Create the Burner Inlet and Tube Coil Walls”](#) in the User's Guide



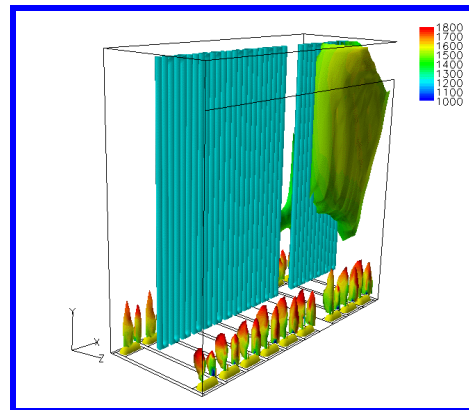
How to create contours on a scalar surface

See “Step 3: Create Filled Contours on a Scalar Surface” in the User's Guide

**How to use Iso-Surfaces to visualize the combustion process**

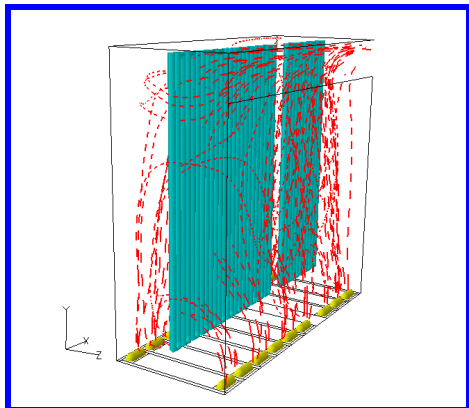
See “Step 4: Visualize the Flame Envelope” in the User's Guide

See “Step 5: Visualize the Combustion Process” in the User's Guide

**Using streamlines to investigate air flow patterns and gas temperature**

See “Step 6: Use Streamlines to Visualize the Flow Pattern” in the User's Guide

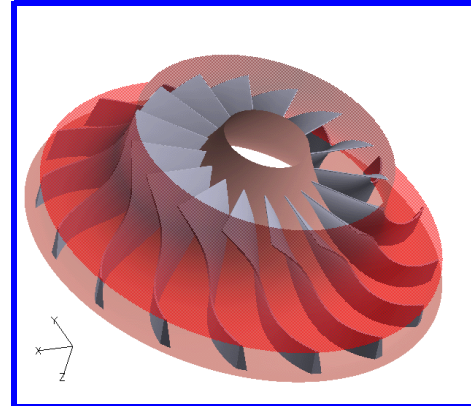
See “Step 7: Visualize the Gas Temperature” in the User's Guide



Working with Turbomachinery Data

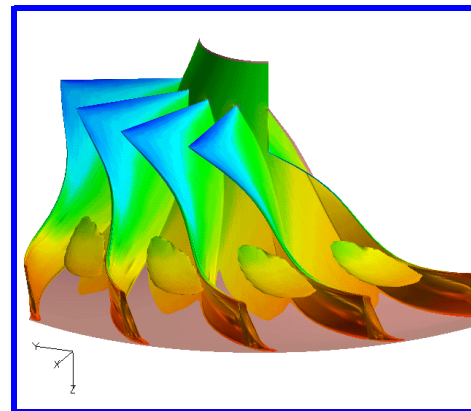
How to rotate a dataset with periodic boundary conditions

See “Step 1: Working with Turbomachinery Data” in the User's Guide

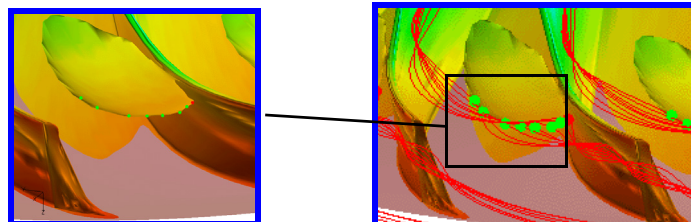


How to use an iso-surface to show flow inefficiencies

See “Step 2: Use an Iso-Surface to Show Flow Blockage” in the User's Guide

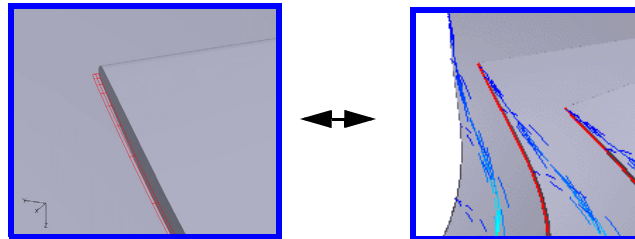


Using an iso-surface for streamline seeding *and* How to display streamlines downstream and upstream of the streamline seeds

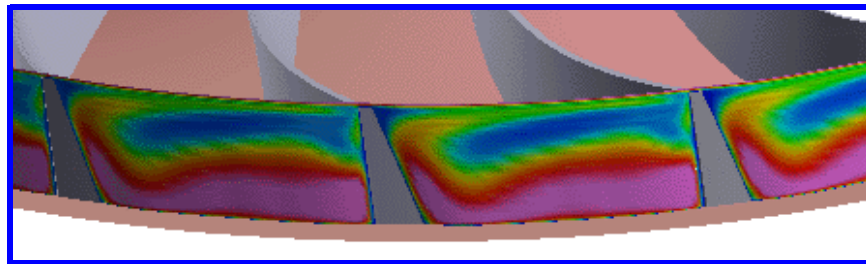


**Visualizing Flow Leakage and
How to display streamlines as particle paths**

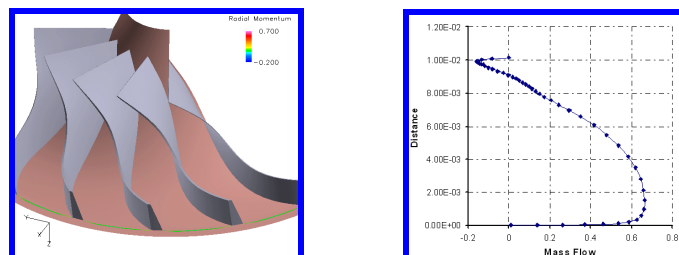
See [“Step 4: Visualize Flow Leakage”](#) in the User's Guide

**How to use the Function Panel to display radial pressure contours and
Creating and “Quick-Picking” the annotation**

See [“Step 5: Create Radial Pressure Contours”](#) in the User's Guide

**How to use “Sweep Integration” to calculate the mass flow rate and displaying the results in a
2D plot.**

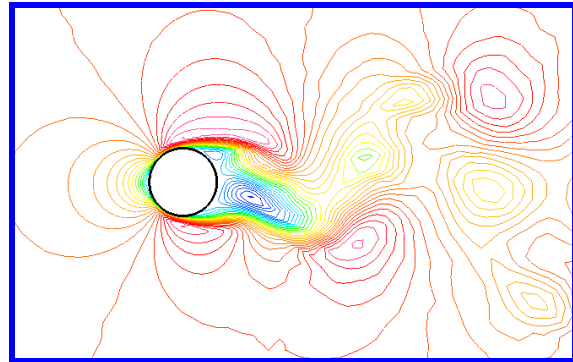
See [“Step 6: Radial Momentum at the Blade Passage Exit”](#) in the User's Guide



Working with Transient Data

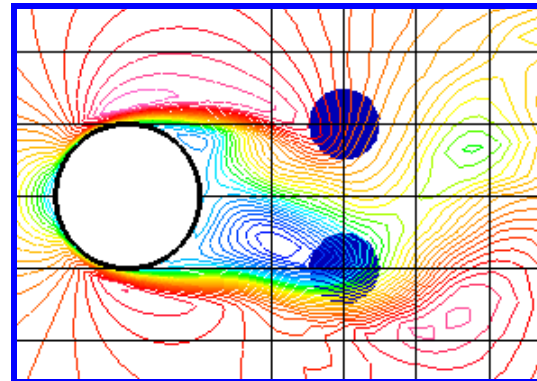
How to read in a FieldView Unstructured Transient Data File *and* Animating the Vortex Tail

See “Step 2: Perform a Transient Sweep” in the User's Guide



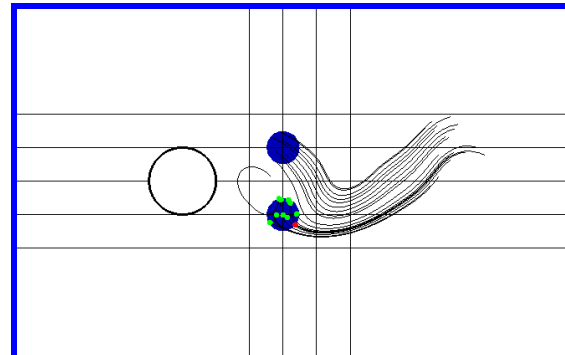
How to create streamline seeding surfaces

See “Step 3: Create Streamline Seeding Surfaces” in the User's Guide



Creating Streamlines and setting the Streamline Time Limit

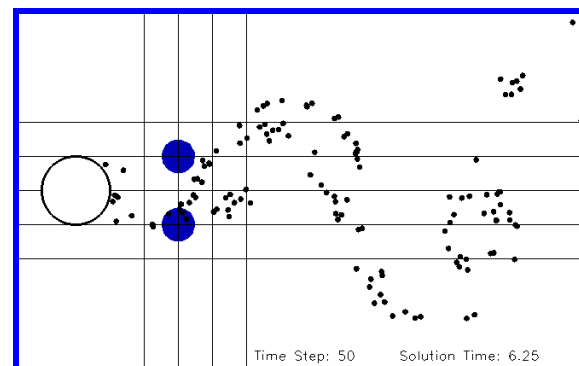
See “Step 4: Create Streamlines” in the User's Guide



Creating Streaklines *and* How to make a movie of a transient streakline *and* How to display the time step and solution time of a transient sweep

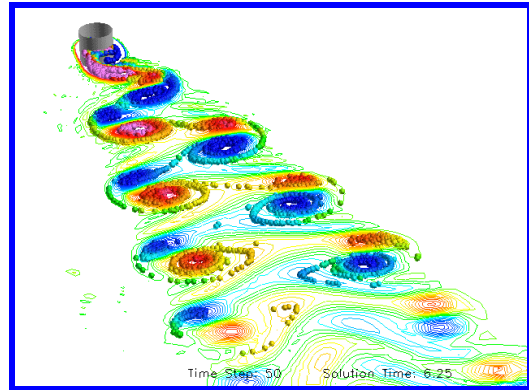
See “Step 5: Create Streaklines” in the User's Guide

See “Step 6: Import Streaklines and Improve your Animation” in the User's Guide



How to use Scripts and the DataGuide™ to improve your animation

See “Step 7: Using Scripts and DataGuide™” in the User's Guide

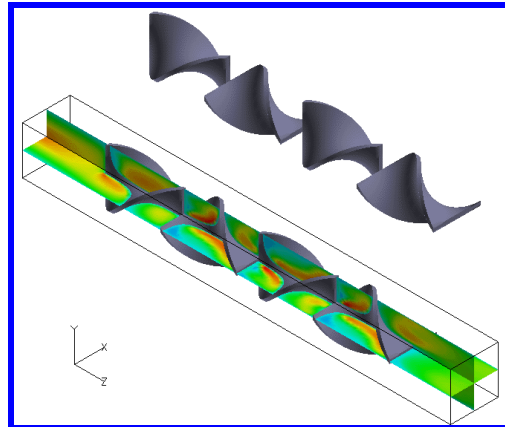


Working with Polymer Processing Data

Creating Boundary Surfaces on Unstructured Data *and*

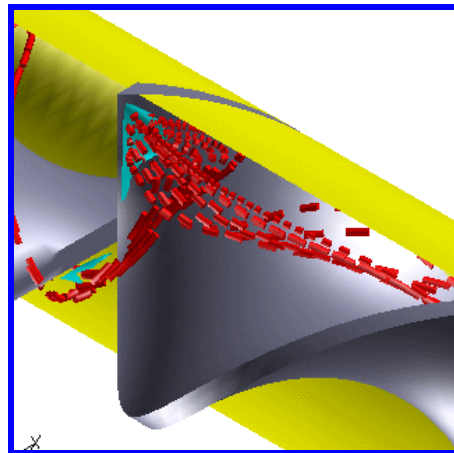
How to increase colormap contours

See [“Step 1: Create Boundary Surfaces”](#) in the User's Guide



Visualizing the Back Flow using Streamlines

See [“Step 2: Visualize Back Flow”](#) in the User's Guide

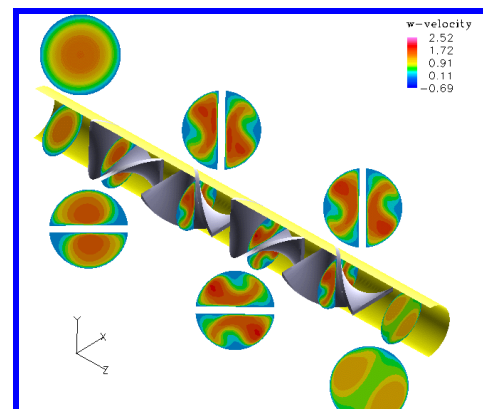


How to create multiple cross sections to visualize axial velocities *and*

Simultaneously viewing axial velocity cross sections alongside the mixer *and*

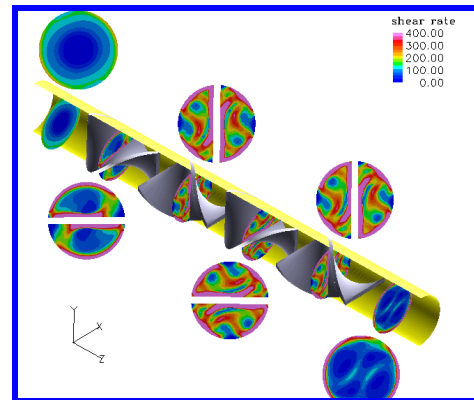
Creating a legend and change the number of decimal places of the displayed value

See [“Step 3: View Axial Velocities”](#) in the User's Guide

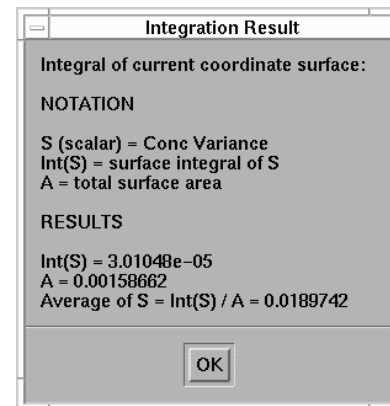


**How to display shear rates on blade surfaces and
How to import a Restart File**

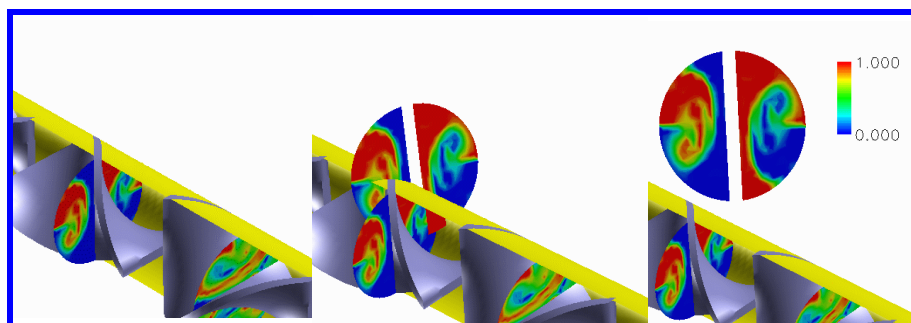
See “Step 4: Change the Scalar Display of the Cross Sections” in the User's Guide

**How to integrate the “current” surface of concentration
variance**

See “Step 5: Integrate Concentration Variance of Each
Cross Section” in the User's Guide

**Using Keyframe Animation and Restarts to translate surfaces and create a “fade-in”**

See “Step 6: Create a Keyframe Animation” in the User's Guide



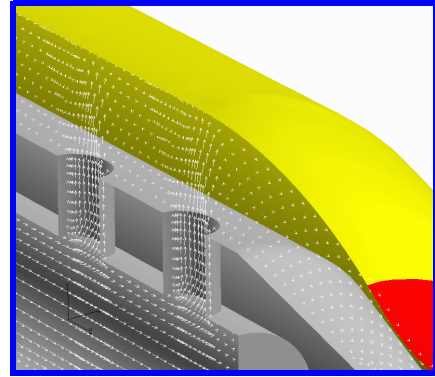
Working with Biomedical Device Data

How to append two datasets

See [“Step 1: Append Two Datasets”](#) in the User's Guide

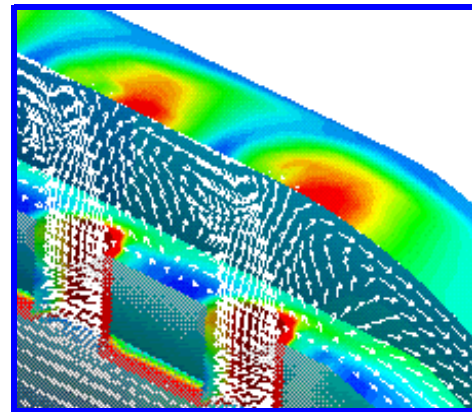
How to Visualize recirculation zones *and* Two datasets in one sessions

See [“Step 2: Visualize Recirculation Zones”](#) in the User's Guide



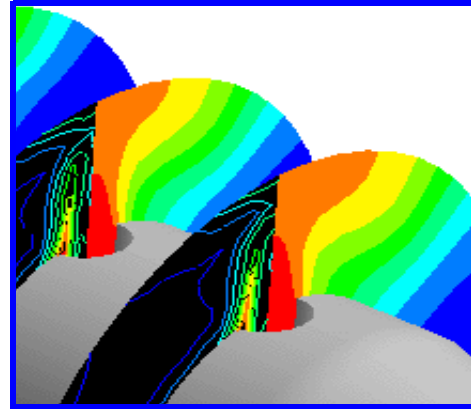
How to visualize the shear on an artery wall *and* How to apply characteristics of one surface to all surfaces of the same type using Unify.

See [“Step 4: Visualize Stress and Concentration Contours”](#) in the User's Guide



How to visualize the stress and concentration contours near the location of the catheter holes

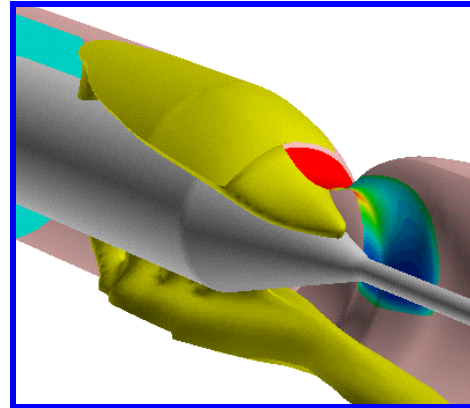
See [“Step 4: Visualize Stress and Concentration Contours”](#) in the User's Guide

**Calculating the Mass Balance**

See [“Step 5: Calculate the Mass Balance”](#) in the User's Guide

Visualizing the flow of the medicine

See [“Step 6: Visualize the Drug Delivery”](#) in the User's Guide



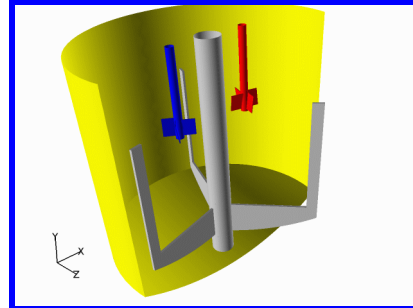
Working with Mixing Models

How to read in CFX-4 Data

See “Step 1: Read in FV-UNS Data” in the User's Guide

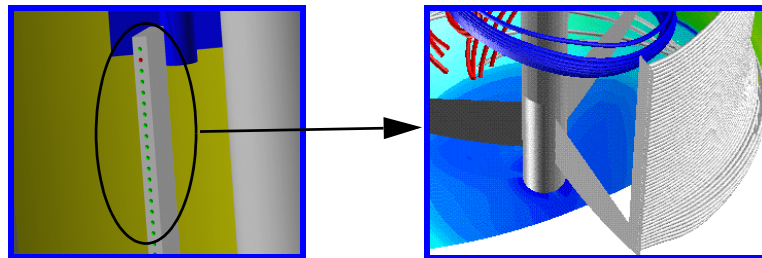
How to Analyze flow in Shafts

See “Step 2: Analyze Flow around the Rotating Shafts” in the User's Guide



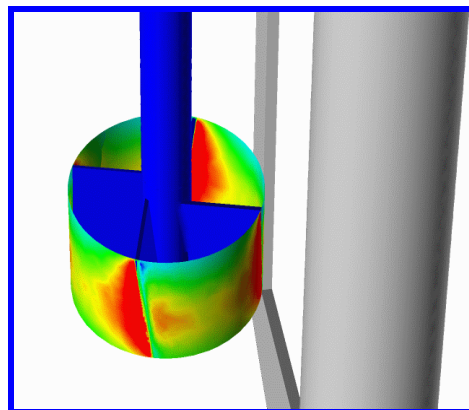
How to automatically add streamline seeds between two seed points

See “Step 3: Examining Mixing Patterns” in the User's Guide



How to calculate the impeller flow number *and* How to increase integration accuracy

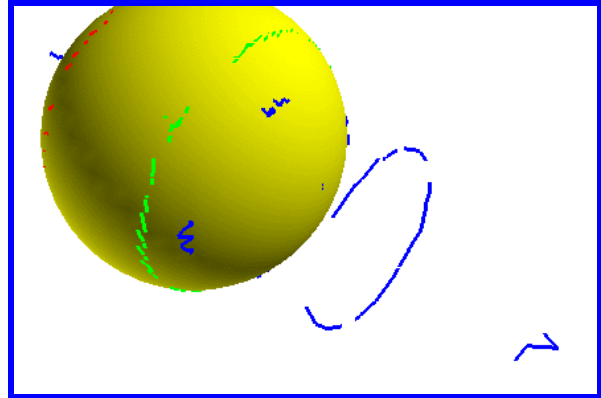
See “Step 4: Use the Function Calculator to Compute the Flow Number” in the User's Guide



Working with Feature Extraction Techniques

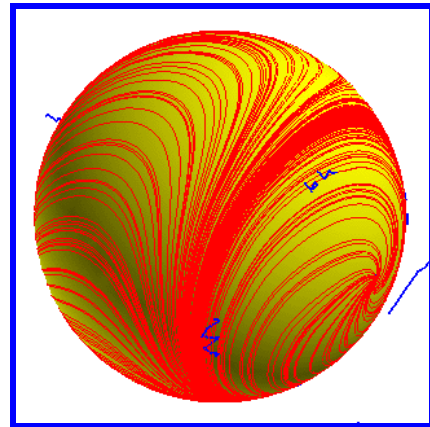
How to automatically display Vortex Cores

See “Step 1: Create Vortex Cores” in the User's Guide



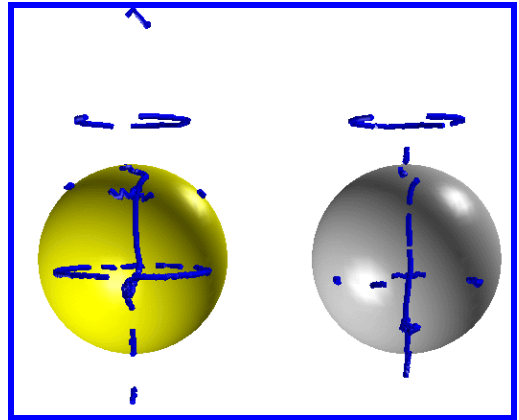
How to automatically display Surface Streamlines Cores

See “Step 2: View Surface Flows” in the User's Guide

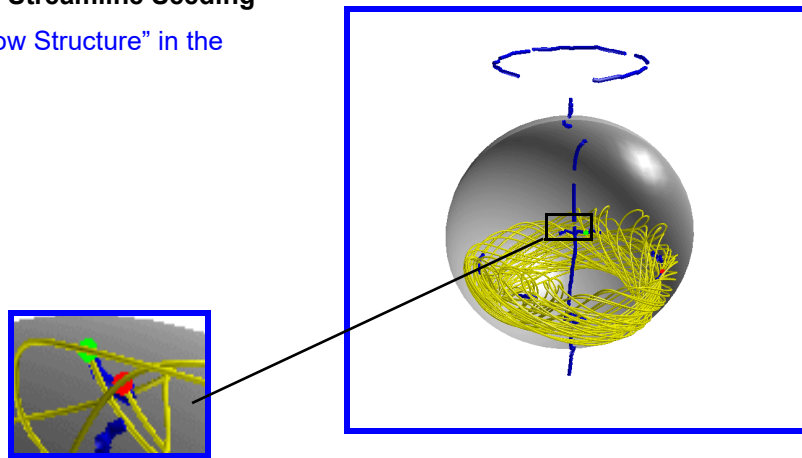


How to analyze two datasets simultaneously

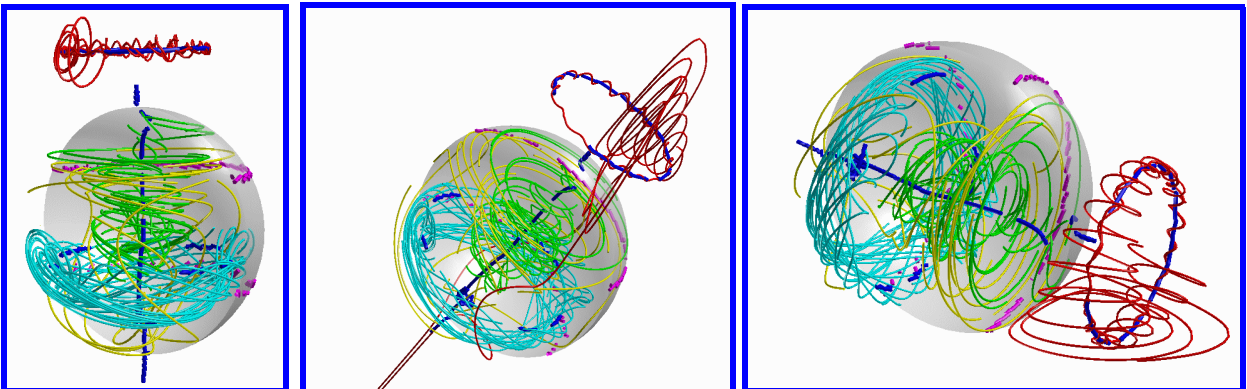
See “Step 3: Analyze Two Datasets Simultaneously” in the User's Guide

**How to use Vortex Cores for Streamline Seeding**

See “Step 4: Show Internal Flow Structure” in the User's Guide

**Create a keyframe animation of the vortex cores**

See “Step 5: Create a Keyframe Animation” in the User's Guide

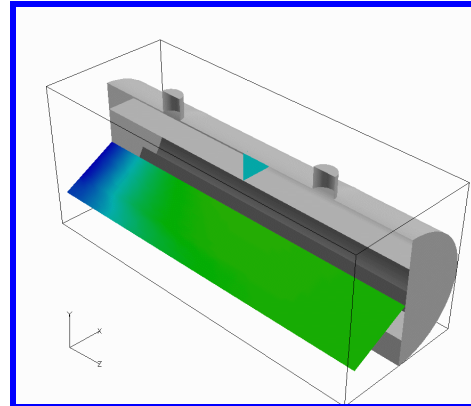


Working with Heat Exchanger Models

How to convert Cartesian data into cylindrical coordinates *and*

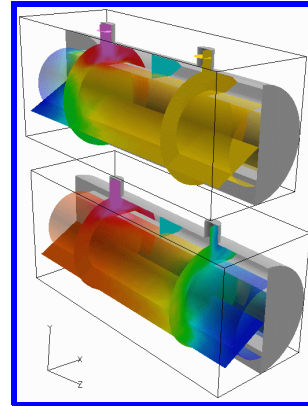
How to create a theta coordinate surface

See “Step 1: Display with Cylindrical Coordinates” in the User's Guide



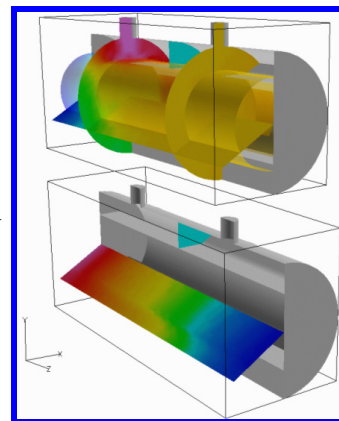
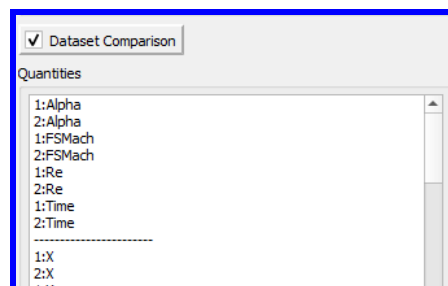
How to automatically create surfaces from previous datasets

See “Step 2: Automatically re-create surfaces from a different dataset” in the User's Guide



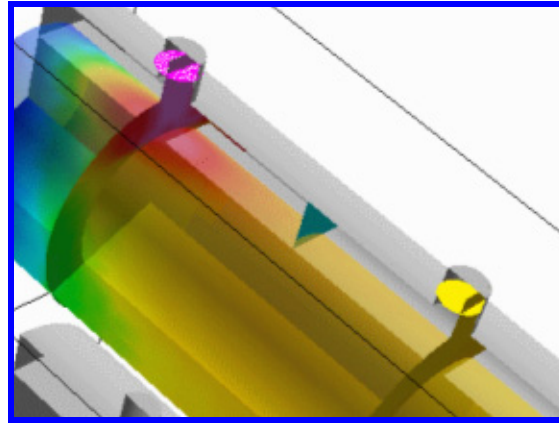
How to compare dataset results

See “Step 3: Compare Datasets” in the User's Guide

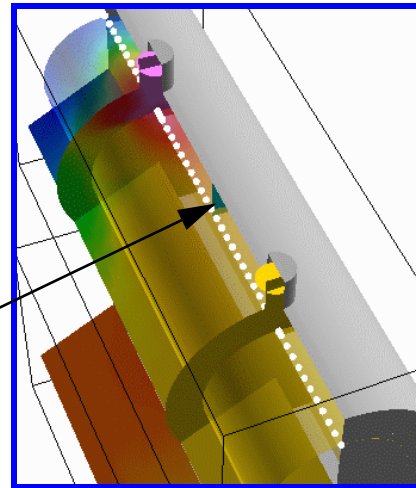
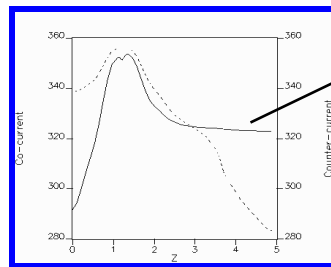


How to use partial Integration

See “Step 4: Use Partial Integration for Convective Flow Calculations” in the User's Guide

**How to create 2-D plots from the intersection of surfaces**

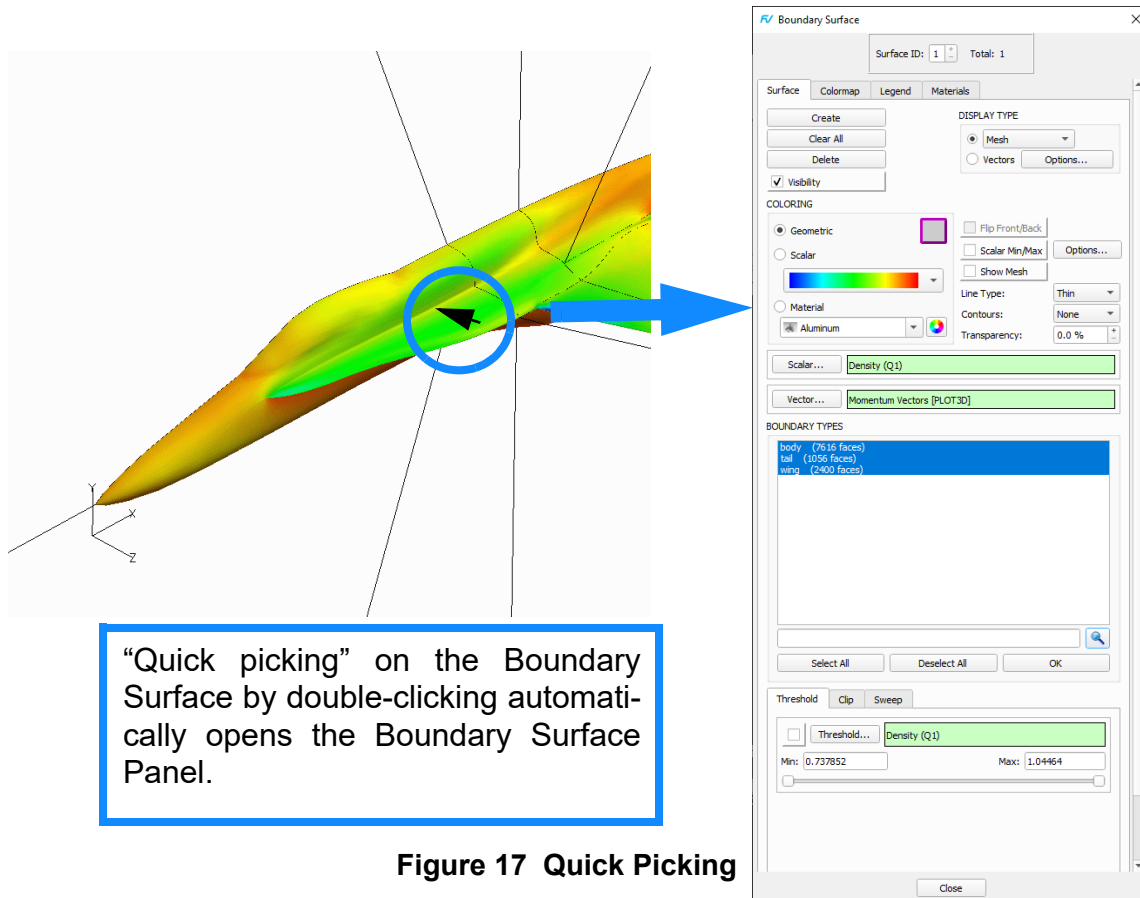
See “Step 5: Create a 2D Plot of Temperature” in the User's Guide

**How to create a KeyFrame animation from multiple datasets**

See “Step 6: Create a KeyFrame Animation using Multiple Datasets” in the User's Guide

Quick Pick

To navigate to a surface or rake in your display, you can open a visualization panel from either the side toolbar or the Visualization pulldown menu. However, a much easier way to raise the visualization panel for any surface or rake is simply to double-click on it. This technique is called **Quick Pick** and applies to all surfaces and rakes in **FieldView**. The lets you quickly navigate through surface and rake panels. Quick Pick can be used across Datasets and can be used in Keyframe Animation. A good example of using Quick Pick would be in editing surfaces as shown below in [Figure 17 Quick Picking](#) and [Figure 18 Streamlines Formation Below Engine Valve](#).



Quick Pick to Reduce Streamline Clutter

When a rake is picked, the seed associated with the selected streamline is made current. An example would be if a streamline rake has been created and you want to reduce the clutter as shown in the pictures below. This feature allows the user to delete individual streamlines from the rake.

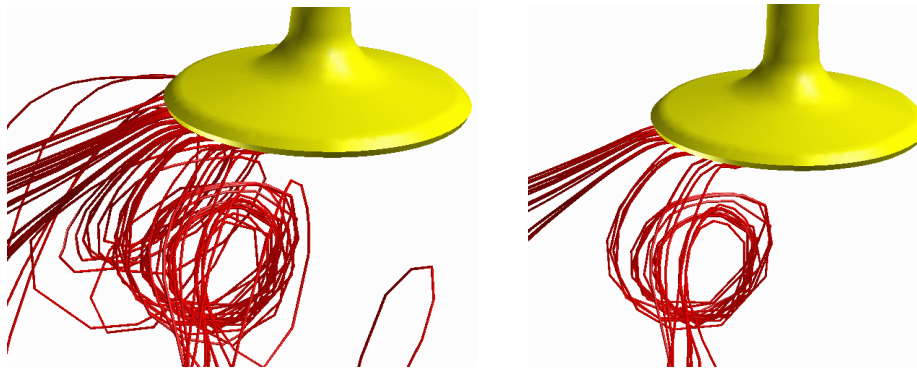


Figure 18 Streamlines Formation Below Engine Valve

As can be seen, the vortex formation on the right is more clearly visible than in the picture on the left. The reduction in clutter was accomplished by:

1. Double-click with the left mouse button (M1). The Streamline panel automatically opens and the Seed Number is displayed in the Seed sub-menu.
2. Changing the Mode to Delete.
3. Clicking on the OK button. The Seed and the Streamline are deleted.

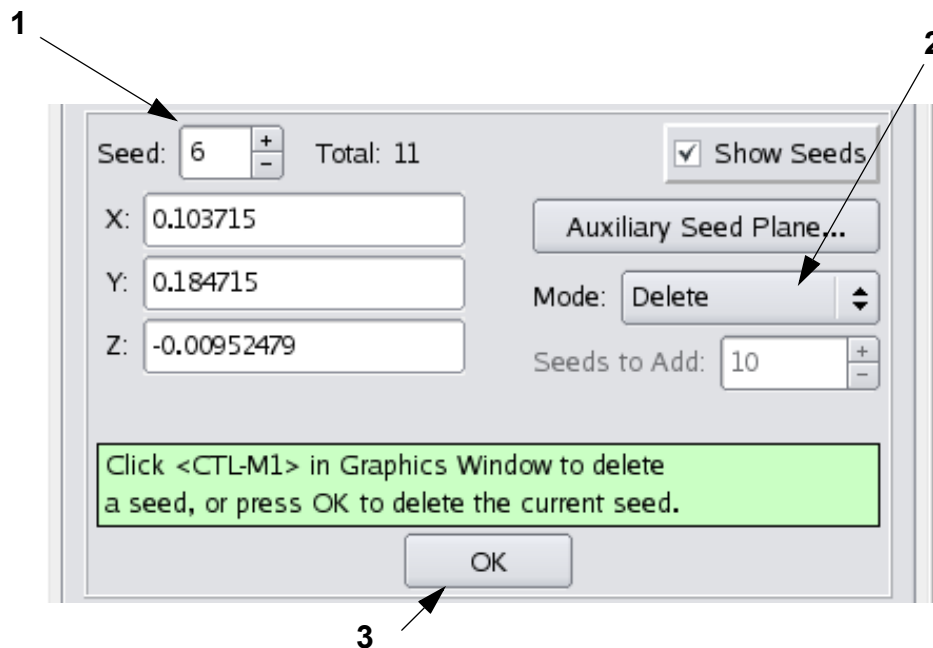


Figure 19 Deleting Streamlines

Restart Files

Due to the complexity involved in the formulas for some of the tutorials, Formula Restart files have been provided to you and should be used when instructed during the tutorials. These restart files contain the names and the mathematical formulation of the equations used. Information about formula restarts can be found in [Chapter 5](#) of the **Reference Manual**. The following formula restart files have been provided in the appropriate tutorial directory containing the data: i) `bio.frm`, ii) `polymer.frm`, iii) `turbo.frm` and iv) `chem.frm`.

Note: To avoid undue complexity of the formulas, their terms and factors have been created through intermediate steps. This “populates” the Function Selection panel with many more formulas than used for a given tutorial.

In addition to the formula restarts files mentioned above, a coordinate surface restart file has been provided to enable you to more easily create the complicated views of the static mixer in the Polymer Processing tutorial (see [Chapter 9](#)). This restart is called `polymer.crd` and can be used to recreate the images with the 6 coordinate surfaces which have been surface transformed away from the dataset.

FVX Tutorial Scripts

FVX scripts have been generated for most of the **FieldView** tutorials. The intent of these scripts is to demonstrate the basic use of **FVX** to read datasets, create surfaces, and perform calculations similar to those actions performed in the **FieldView** tutorials. In some cases, **FieldView** restarts have been used to re-create the steps illustrated for each of the tutorial cases.

To test out these **FVX** scripts, we recommend that you create a subdirectory or folder. Put the tutorial dataset into this folder, and copy the **FVX** scripts and associated restart files. **FVX** scripts for each tutorial are located within the respective tutorial folder on the **FieldView** DVD and are named according to the following naming convention:

```
sampleXXXXXXfvx.fvx
```

where "XXXXXX" indicates the associated tutorial name.

To run these scripts, you do not need to load the data first: the script will do this for you. Several break points have been introduced into these scripts so that the results of each intermediate step can be viewed. When a break point is reached, the **FVX** script is paused. To advance the **FVX** script from a break point, you need to have access to the console window or the window where you launched **FieldView**. Pressing ENTER will let you continue through to the next break point. Since some of these scripts will generate files or animations, it is important that you have write permissions to these directories.

Note: For maximum utility, try opening an **FVX** tutorial script in a text editor while running the same **FVX** tutorial in **FieldView** to observe how the **FVX** syntax and commands create each tutorial step.

Chapter 3

Basic Aero- space Tutorial



This chapter will focus on an aerospace application, looking at feature detection for external flow around an F18. The data used is courtesy of NASA Langley Research Center.

What You Will Learn

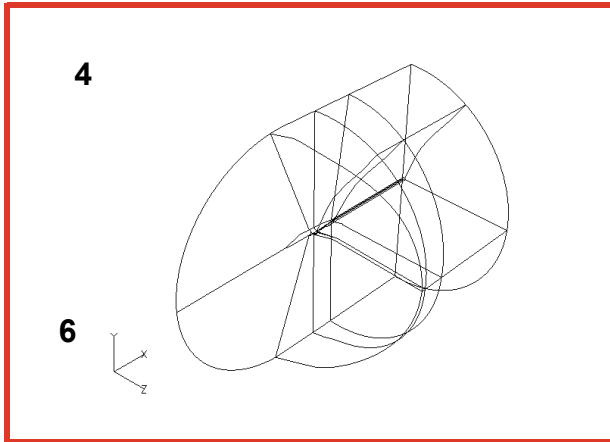
- [How to read a PLOT3D File into FieldView](#)
- [How to create a surface on the boundary of the model using Boundary Surfaces](#)
- [How to Mirror a model](#)
- [How to display velocity vectors using a Coordinate Surface](#)
- [How to create Pressure Profiles](#)
- [How to use the Colormap](#)
- [How to create an Iso-Surface to display a pressure distribution](#)
- [How to create a Transparent Surface](#)
- [How to create Streamlines](#)
- [About visualizing the Mach Number](#)
- [How to create and use Functions for displaying properties](#)
- [How to create a 2D Plot to display properties through a section of the model](#)
- [How to create a complex animation using Keyframe Animation](#)

Step 1: Read the PLOT3D Tutorial Data

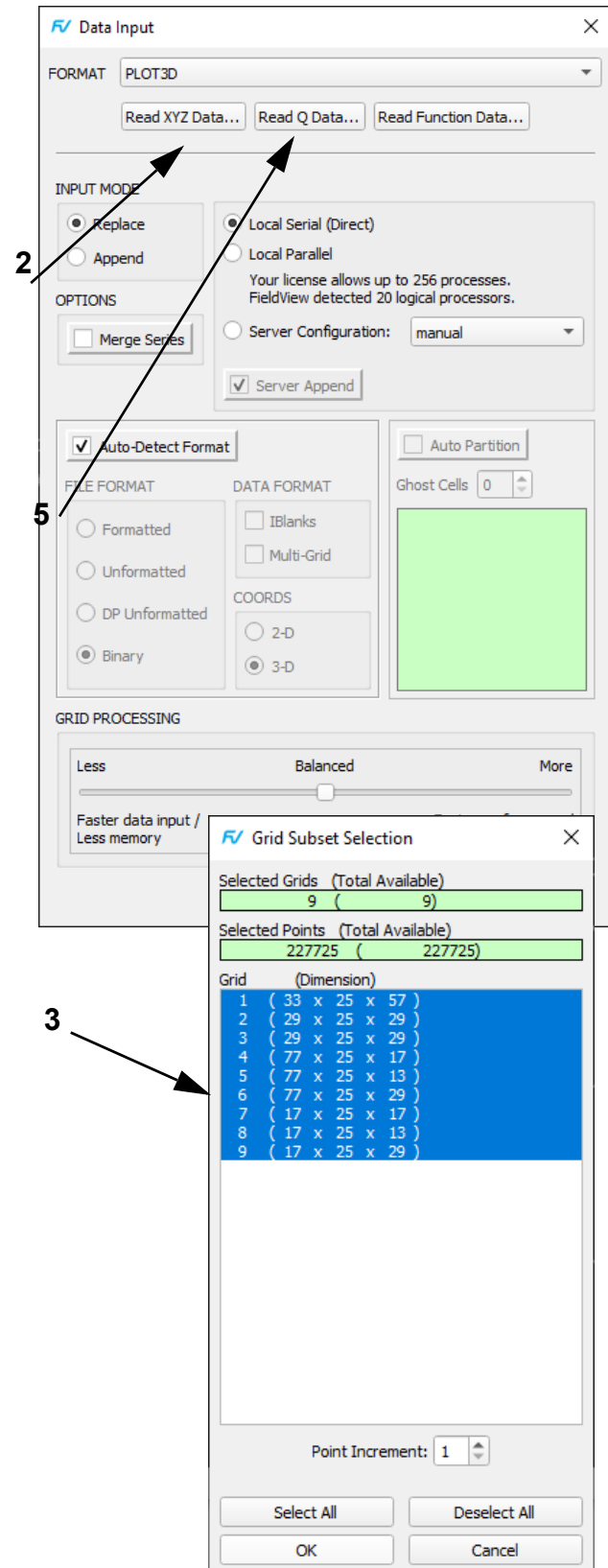
In this step we will create the geometry of the F18 with Boundary Surfaces.



The pictures on each page correspond to the final image you should see when the steps are completed correctly.

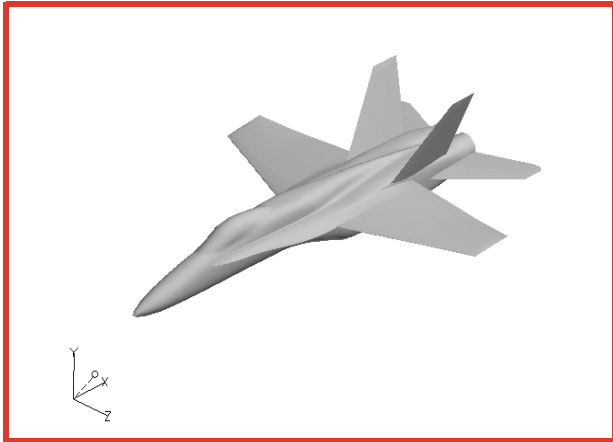


1. Bring up the **File**→**Data Input**→**PLOT3D** Data reader. [\[Page 55 Reference Manual\]](#)
2. Read in the grid data by clicking on **Read XYZ Data...** and browsing for the grid file called **f18i9b_g_bin**. Select this file and press **Open**.
3. The **Grid Subset Selection** panel will appear, showing the 9 grids of this dataset. By default, all of the grids are selected (see [Chapter 1](#) of the **Reference Manual**). **Point Increment** can be changed to subset each grid read in (2 = every other point) but may affect the following visualizations. Press **OK** to select all 9 grids.
4. After the grid data is read in, the outline will appear in the graphics window.
5. Read in the results data by clicking on the **Read Q Data...** button. Select the **Q** file **f18i9b_q_bin** and press **Open**. **Close** the panel.
6. Using the default mouse controls, use M3 to zoom into the outline and M1 to translate to the approximate position shown above. We will create surfaces, mirror the dataset and eventually turn off the outline in the next step.

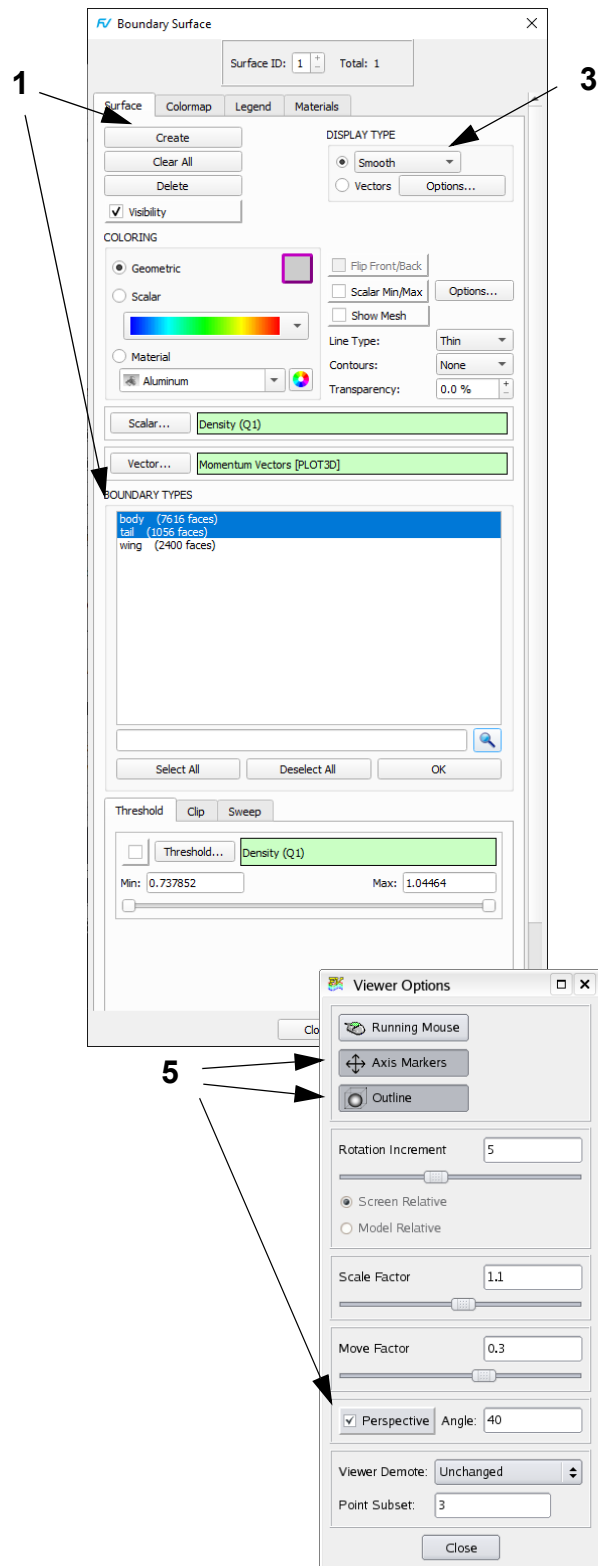


Step 2: Create Boundary Surfaces

In this step we will create the geometry of the F18 with Boundary Surfaces.

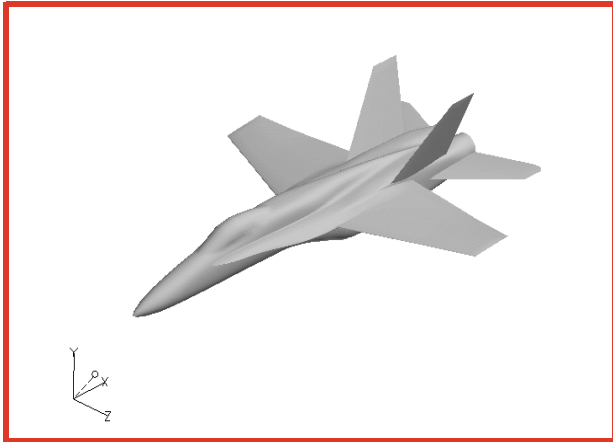


1. Bring up the **Boundary Surface** panel with the Visualization Panels pull-down from the Main Menu or the **Bound** icon on the side toolbar. Press **Create**, and select the body and wing in BOUNDARY TYPES, then press **OK**. They will appear with the default **DISPLAY TYPE** of **Mesh**.
2. Press **Create** again. This time, select the BOUNDARY TYPE tail, then press **OK**. Having separate boundary surfaces will allow us to visualize each one differently later on. [\[Page 241 Working with FieldView\]](#)
3. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading. Select **Unify** on the Tools pull-down on the Main Menu bar to easily display both boundary surfaces the same way. [\[241, 345\]](#)
4. From the Transform Controls toolbar press the Center button. Use the mouse to position (M1) and zoom (M3) the dataset in the graphics window.
5. From the Transform Controls toolbar, click on the **Viewer Options** button. From the **Viewer Options** panel, click the **Outline** to turn its visibility off. **Close** the panel.



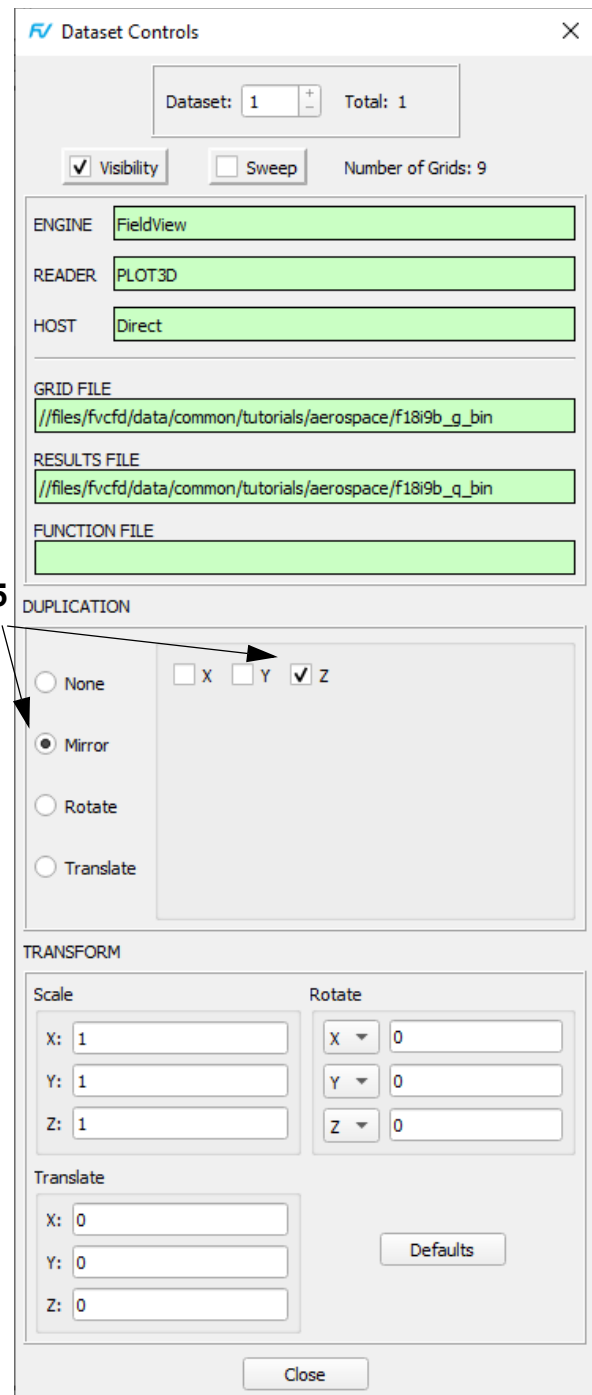
Step 2: Create Boundary Surfaces (cont'd)

Here we show you how to mirror your dataset around a symmetry plane.



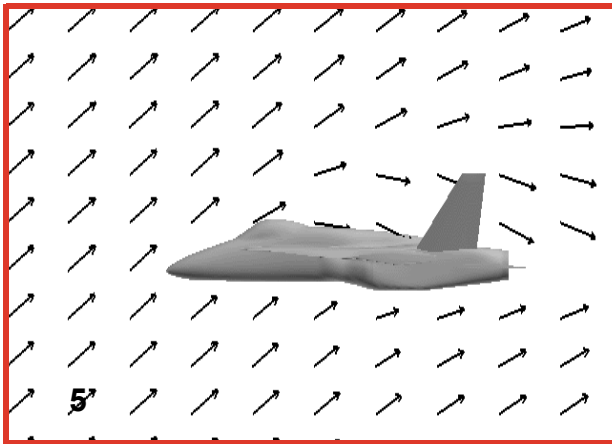
6. From the main toolbar, click on the **Dataset** button to bring up the **Dataset Controls** panel. The panel will display the paths to the GRID (XYZ) and RESULTS (Q) files.
7. Duplicate the dataset by using the **Mirror** option. Turn on the Z button. Then **Close** the **Dataset Controls** panel.

5

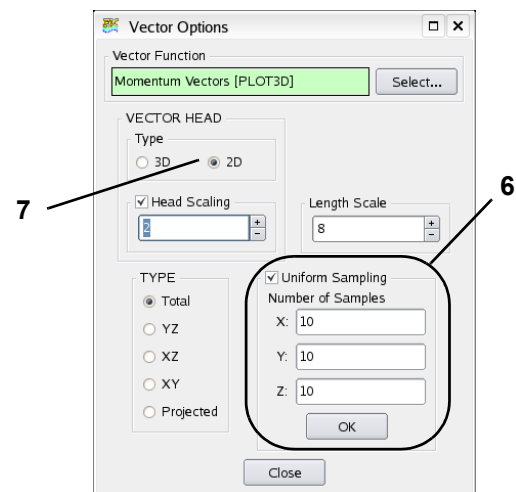
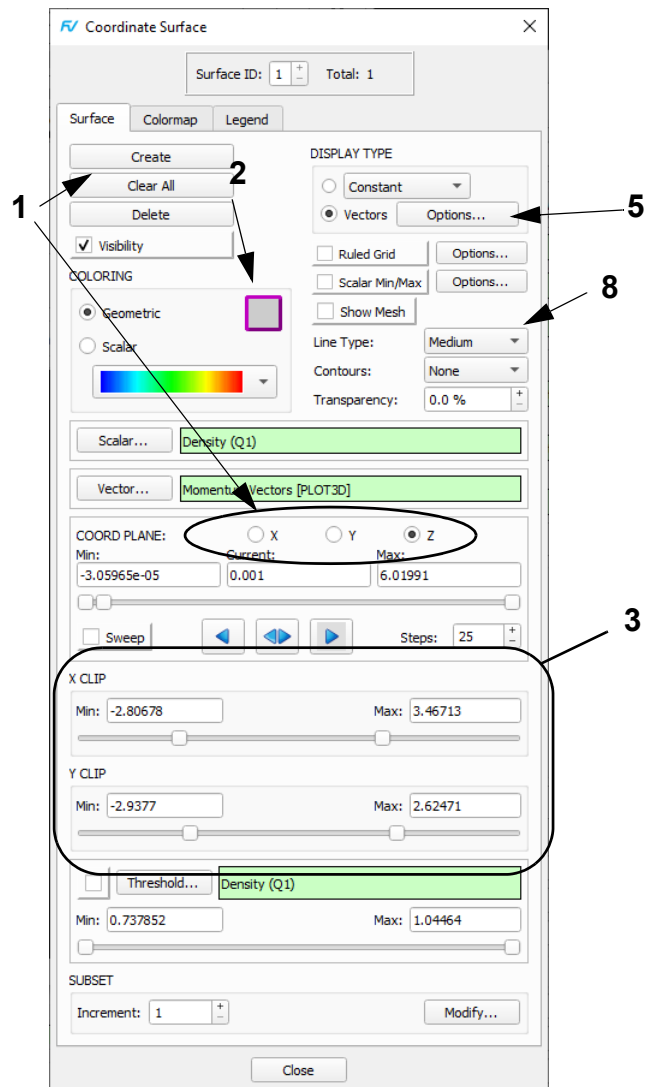


Step 3: Display Velocity Vectors

This step will allow us to visualize the incident angle of attack of the flow field.

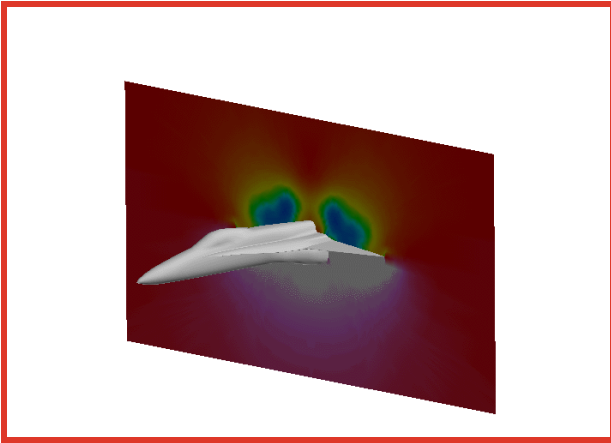


1. Bring up the **Coordinate Surface** panel and **Create** a coordinate surface. Change the COORD PLANE from X to Z. Adjust the **Current** value by entering 0.001 into this field. This will place the Z plane near the center-line of the F18. [\[Page 233 Working with FieldView\]](#)
2. In the **COLORING** section, change the Geometric color from gray to black. This can also be accomplished from the Scalar Colormap Specification panel.
3. In order to better see the surface along with the F18, we will limit the X and Y extent of the Z plane. Move in the **Min/Max** sliders for X and Y to the approximate locations shown on the panel.
4. On the Transform Controls toolbar, click the Defined Views icon. On the **Defined Views** panel click **+Z**. **Close** the **Defined Views** panel. On the Transform controls toolbar, Center, and zoom in a bit with the right mouse button, [M3].
5. Now that we have the surface placed and subsetting the way we want, change the **DISPLAY TYPE** to **Vectors**, then press the **Options...** button.
6. To adjust the **Vector Options** for a better picture. First, turn on **Uniform Sampling**.
7. Note that the default vector head **Type** is **2D**. Turn on **Head Scaling** with a value of **2**, and adjust the **Length Scale** to **8**. This will give us better vector sizes. **Close** the panel.
8. Finally, change the **Line Type** from **Thin** to **Medium** (may not be available on all displays). [\[Page 113 Working with FieldView\]](#)

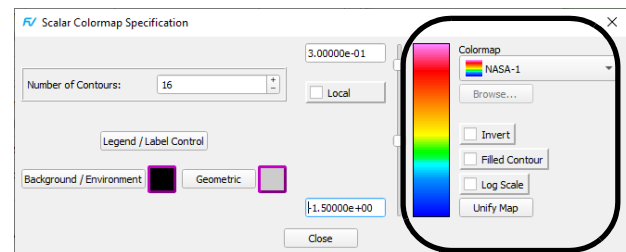
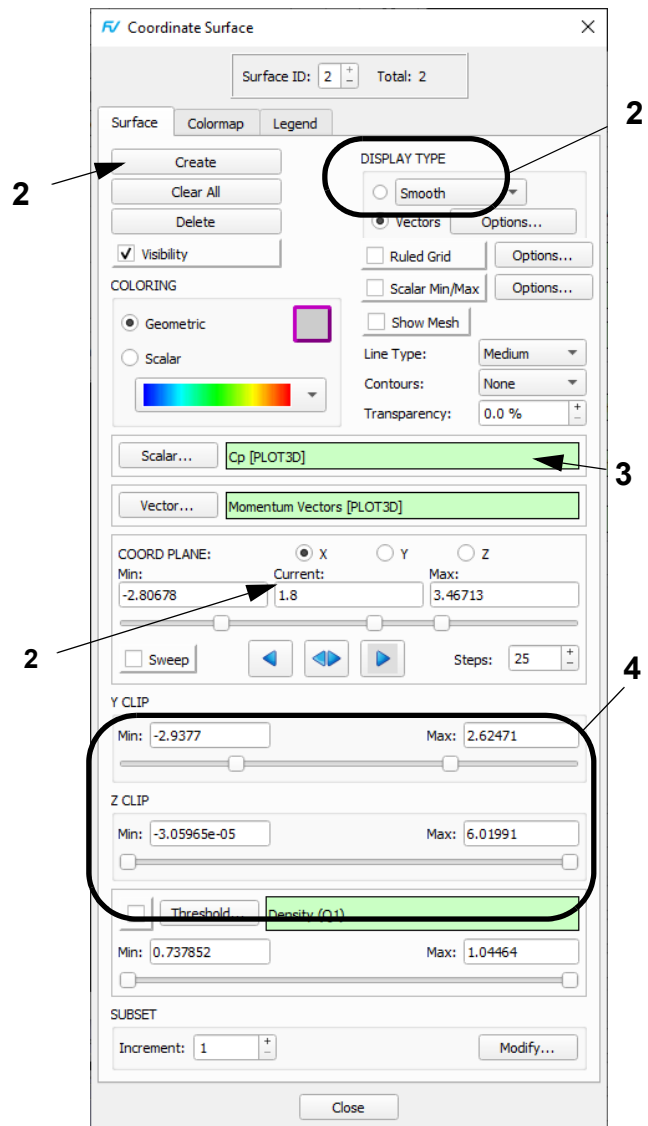


Step 4: Display the Pressure Coefficient

In this step, we will sweep a **Coordinate Surface** of pressure coefficient to help us detect a vortex core. This vortex core will allow us to produce streamlines dramatically demonstrating the effects of the high angle of attack on the flow field.



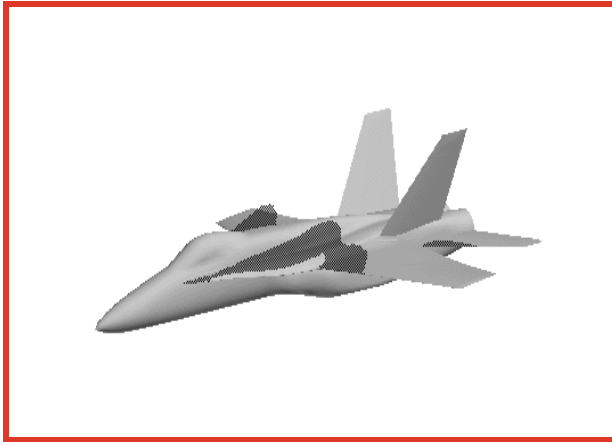
1. To better see the surface we are going to create, turn off the **Visibility** of **Coordinate Surface 1**. On the **Defined Views** panel, orient the model by choosing **-X +Y +Z** in the **ISOMETRIC** section and **Close** this panel. Then zoom and rotate using the mouse controls to approximate the view above.
2. **Create** a second coordinate surface. Change the **DISPLAY TYPE** from **Vectors** to **Smooth** shading and the **COLORING** from **Geometric** to **Scalar**. Turn on **Visibility** and change the **COORD PLANE** from **Z** to **X**. Change the **Current** value to approximately 1.8, and the **Min:** and **Max:** to those shown.
3. To adjust the **Scalar** Function, press the **Select...** button to bring up the **Function Selection** panel (not shown), select **Cp [PLOT3D]** and press **Calculate**. Alternately, you can do this from the **Function Specification** panel. [\[Page 100 Reference Manual\]](#)
4. Clip the surface by adjusting **Min:** and **Max:** in the **Y** and **Z** directions to those shown.
5. Bring up the **Scalar Colormap Specification** panel using the main toolbar button or the **Edit** menu and select Colormap **NASA-2**. Adjust the range to **-1.5** to **0.3** and **Close** the panel.
6. Back on the **Coordinate Surface** panel, turn **Sweep** on. The surface will move from the new **Min:** to the new **Max:** value in 25 equal steps. Turn off **Sweep**. [\[Page 232 Working with FieldView\]](#)



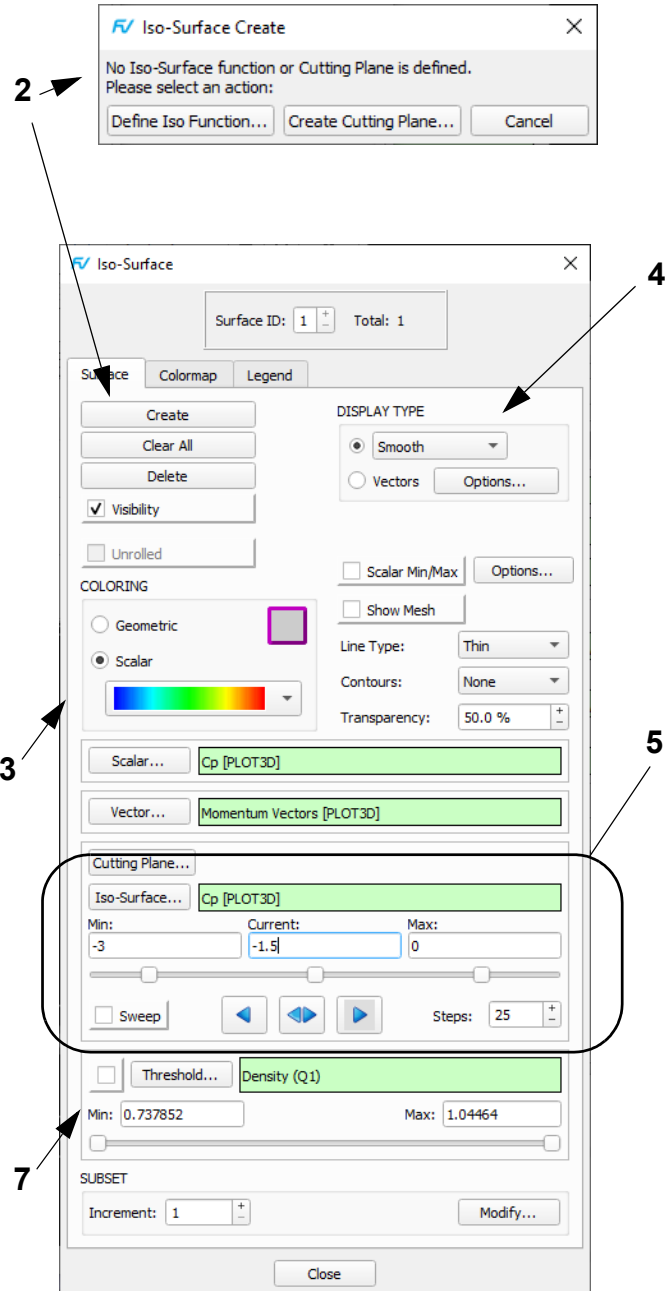
5

Step 5: Visualize the Vortex Core

In this step, we will visualize the vortex core with an Iso-Surface. We will then use this Iso-surface as a streamline seeding surface that will allow us to precisely locate and position streamlines.

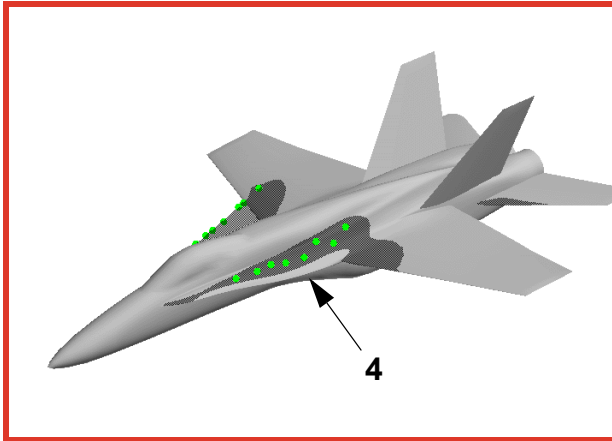


1. To better see the iso-surface we are going to create, turn off the **Visibility** of **Coordinate Surface 2** (panel not shown).
2. Bring up the **Iso-Surface** panel using the **Iso** icon on the side toolbar and press the **Create** button. Since this is the first iso-surface we are creating, we will be prompted to select the iso-function we want to use. Click on the **Define Iso Function** button, select **Cp [PLOT3D]** from the **Function Selection** panel (not shown) and **Calculate**. The **Min:** and **Max:** values of the **Iso Function** slider will be the range of values of **Cp [PLOT3D]**. Temporarily, move the **Current** value slider to -1.5. [\[Page 171 Working with FieldView\]](#)
3. Change the **COLORING** from **Geometric** to **Scalar**.
4. Change the **DISPLAY TYPE** from **Constant** to **Smooth** shading.
5. Adjust the **Min:** and **Max:** of the **Iso Function** to -3 and 0, respectively. This will give us a good range of values to sweep through.
6. So that we won't hide the F18 with the surface, let's set the **Transparency** to **50%**.
7. Press the **Sweep** button to see the surface sweep from the set **Min:** to the set **Max:** value.
8. Turn off the **Sweep**. Set the **Current** value to -1.5. This will give us a good surface to seed for streamline creation in the next section (action not shown).

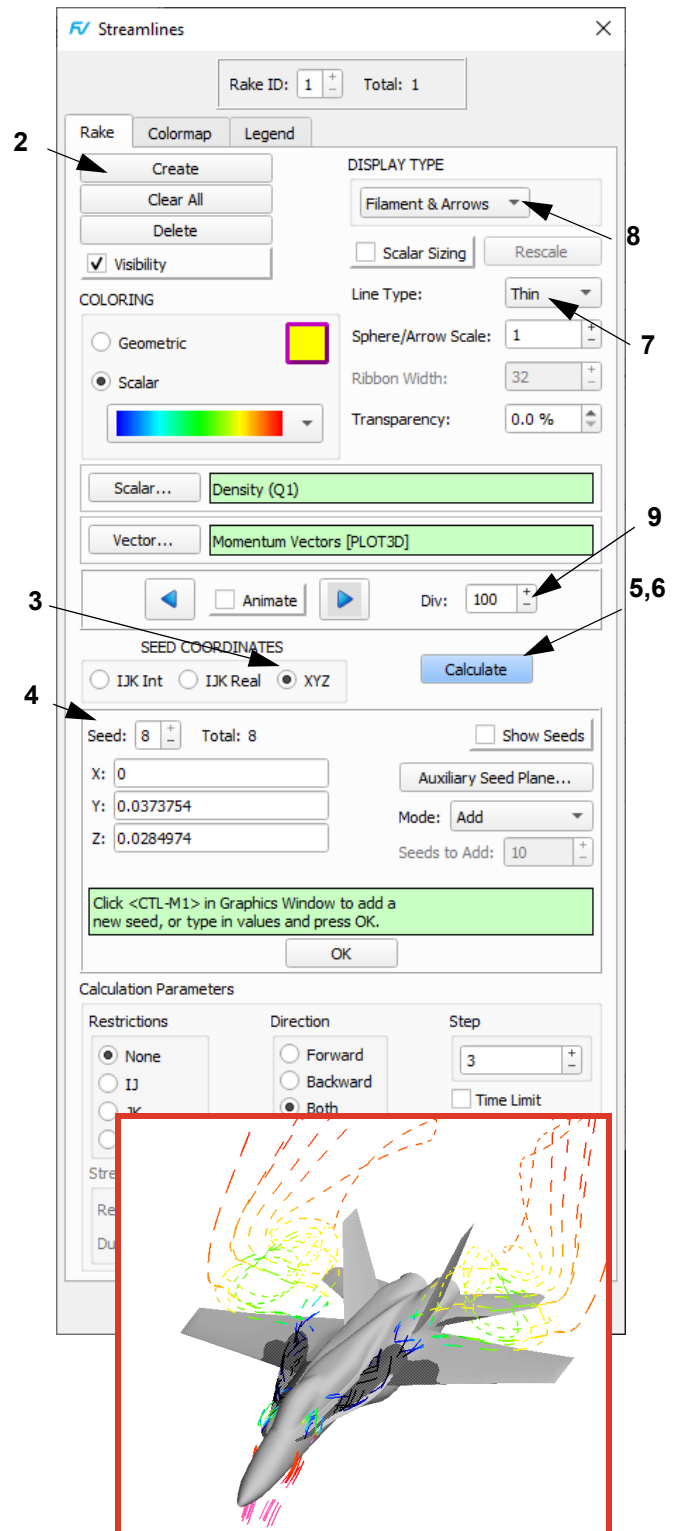


Step 6: Create Streamlines

We will now use the Iso-surface of **Cp [PLOT3D]** equal to **-1.5** as a seeding surface for streamlines. To better see the Iso-surface for seeding, rotate down and around a bit with M2 and zoom in with M3.

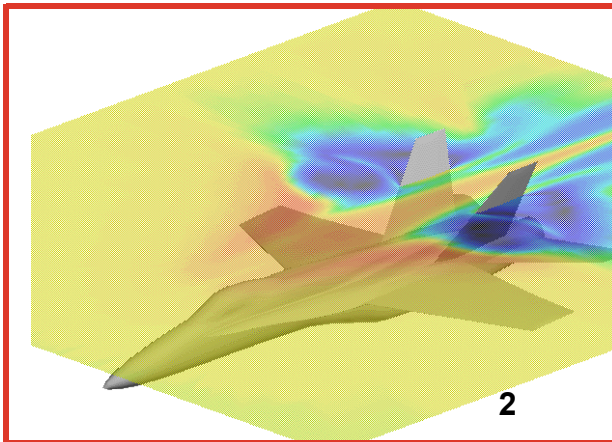


1. Orient the model down and zoom in. Then bring up the **Streamlines** panel. [\[Page 184 Working with Field-View\]](#)
2. Press the **Create** button on the **Rake** tab.
3. Change the **SEED COORDINATES** from **IJK Int** to **XYZ**. The default seeding **Mode: Add** will be sufficient for our usage here.
4. Using the Control key on the keyboard in conjunction with M1, place 8 seeds on the Iso-surface with the mouse. You can place the seeds in any pattern along the surface. We have chosen the locations shown. Note that since our dataset is mirrored, everything that appears on the port side (facing us) also appears on the starboard side. We will limit our placing to the port side.
5. Press the **Calculate** button to compute the streamlines. They will appear with their default **DISPLAY TYPE** of **Complete** and default integration **Direction** of **Forward**. Change the **COLORING** from **Geometric** to **Scalar**.
6. In the **Calculation Parameters** section of the **Streamlines** panel (near bottom), adjust the **Direction** to **Both**. Press **Calculate** again on the **Streamlines** panel.
7. Change the **Line Type** to **Medium**.
8. Change the **DISPLAY TYPE** to **Filament & Arrows**.
9. Change the number of divisions **Div:** to **100** and Enter. Turn off **Show Seeds**.
10. Turn on the **Animate** button. To get the picture at right, the view can be rotated as desired. Turn off the **Animate** button.

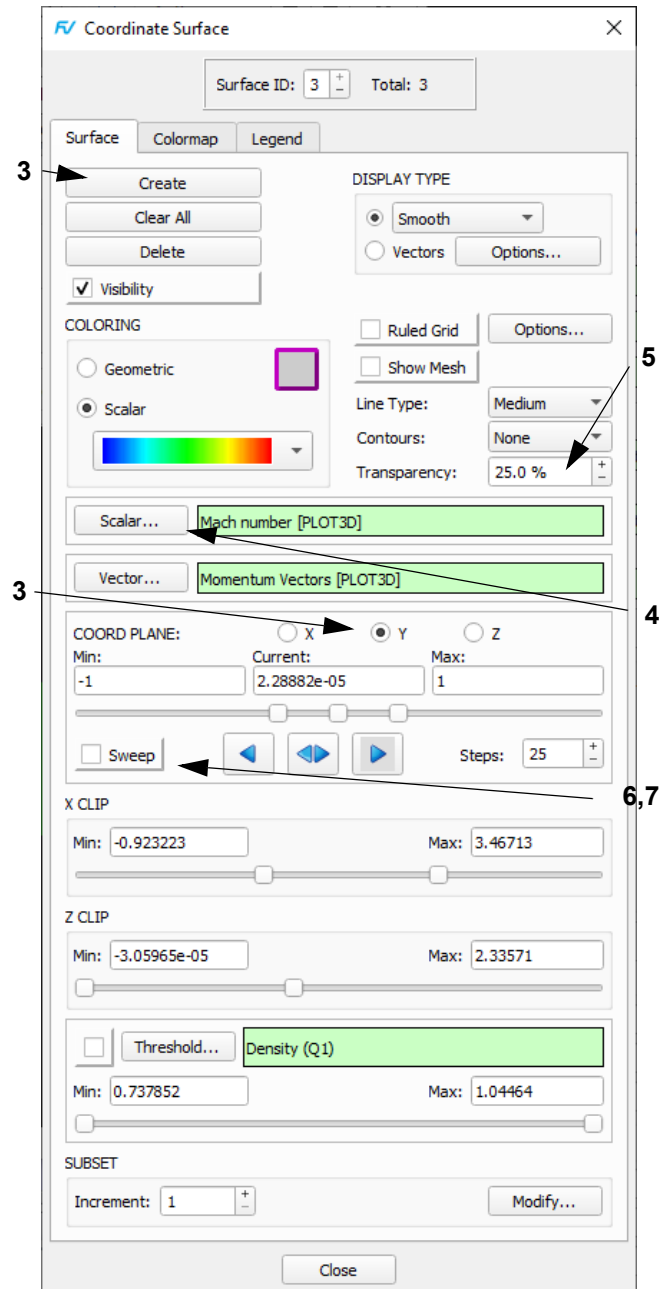


Step 7: Display the Mach Number

We will now examine the Mach number in the vicinity of the tail by sweeping a coordinate surface.

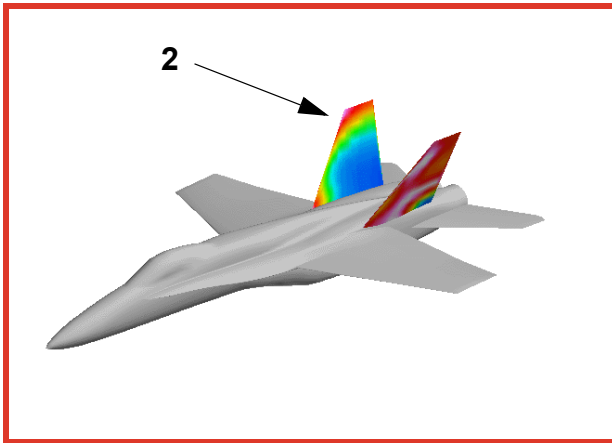


1. Turn off the **Visibility** of the **Streamlines** and the **Iso-surface** (panels not shown).
2. Orient the model similar to that shown.
3. **Create** a third **Coordinate Surface**. Turn **Visibility** on. Change the **COORD PLANE** from **X** to **Y**.
4. Press the **Scalar...** button for the **Scalar Function**. Choose **Mach Number[PLOT3D]** from the **Function Selection** panel and **Calculate**. This selection can also be accomplished using the **Function Specification** panel. [\[Page 100 Reference Manual\]](#)
5. Set the **Transparency** to **25%**.
6. Press **Sweep** on the **Coordinate Surface** panel.
7. When desired, turn off **Sweep** and set the **Current** value to **0.33333**. You can do this by pressing the left or right arrows to step to the correct value, or by typing the value into the field and pressing Enter on the keyboard.

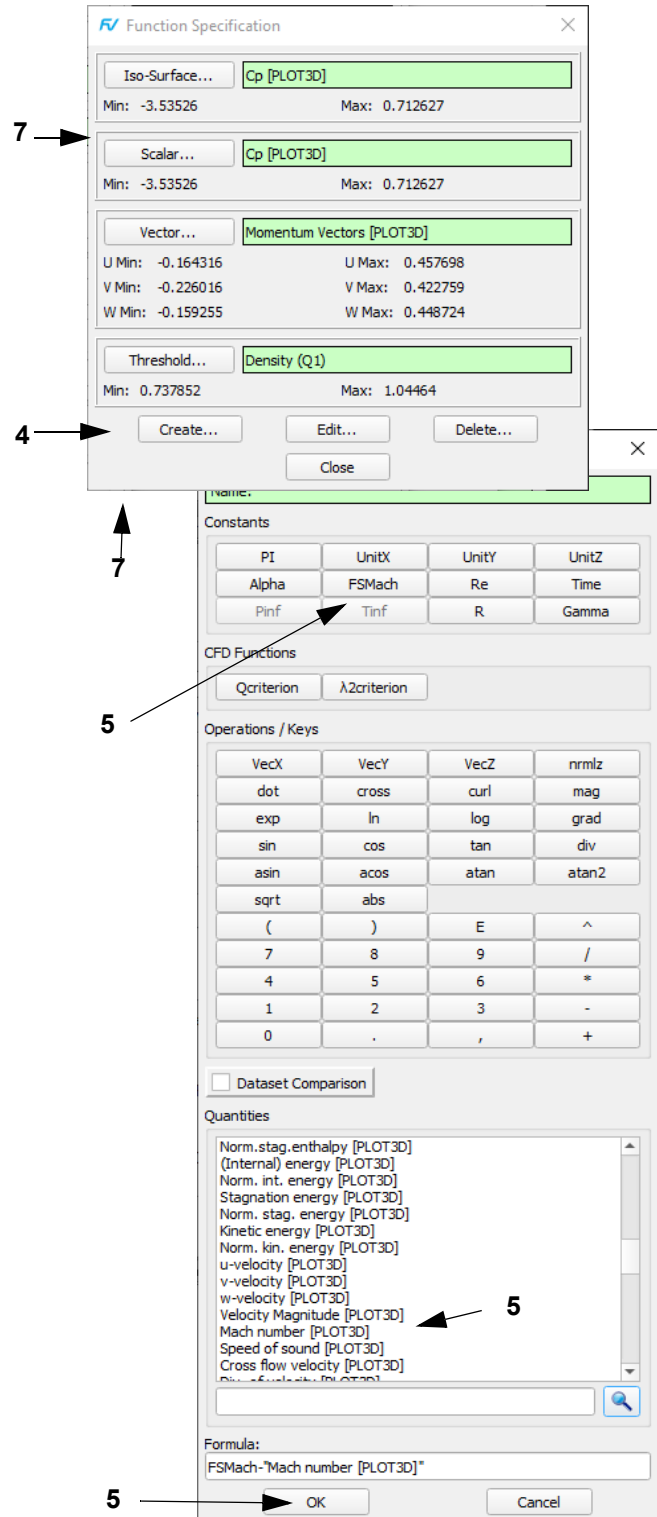


Step 8: Create a Scalar Function

We will now display the tail of the F18 colored with a scalar variable, which we created as a separate Boundary Surface earlier, with a function we will create with the Function Formula Specification panel ("Function Calculator").

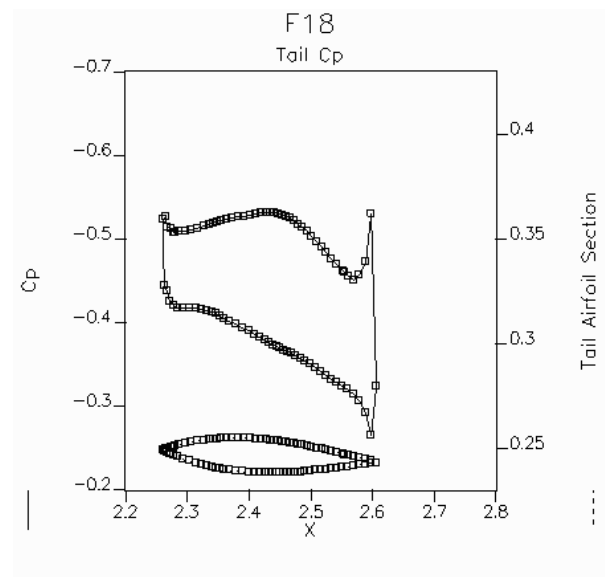
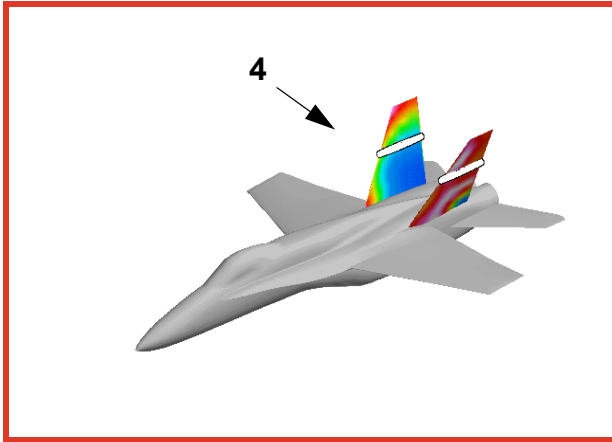


1. Turn off the **Visibility** of **Coordinate Surface 3** (panel not shown).
2. Double-click or "Quick-Pick" on the tail of the F18 to bring up the **Boundary Surface** panel.
3. Change the **COLORING** to **Scalar**. The current **Scalar Function** is **Mach Number [PLOT3D]**. We will now change that to our own defined function.
4. Press the Functions button on the main toolbar. Press **Create...** to bring up the **Function Formula Specification** interface. We will create our function here. [\[Page 100 Reference Manual \]](#)
5. Create the function which is the free-stream Mach number minus the local Mach number. Do this by pressing the **FSMach** button, the minus sign, then choose **Mach Number [PLOT3D]** from the **Quantities** list. Then press **OK**.
6. When the **Function Name Input** panel comes up, press **Use Formula**, which will give the formula the same name as the formula statement itself (panel not shown).
7. Press the **Scalar...** button on the **Function Specification** panel. Scroll to the bottom of the list of functions on the **Function Selection** panel and choose **FSMach - "Mach number [PLOT3D]"**. Press **Calculate**. **Close** the **Function Specification** panel. Change the range to -0.1 to 0.3 on the **Scalar Colormap Specification** panel.

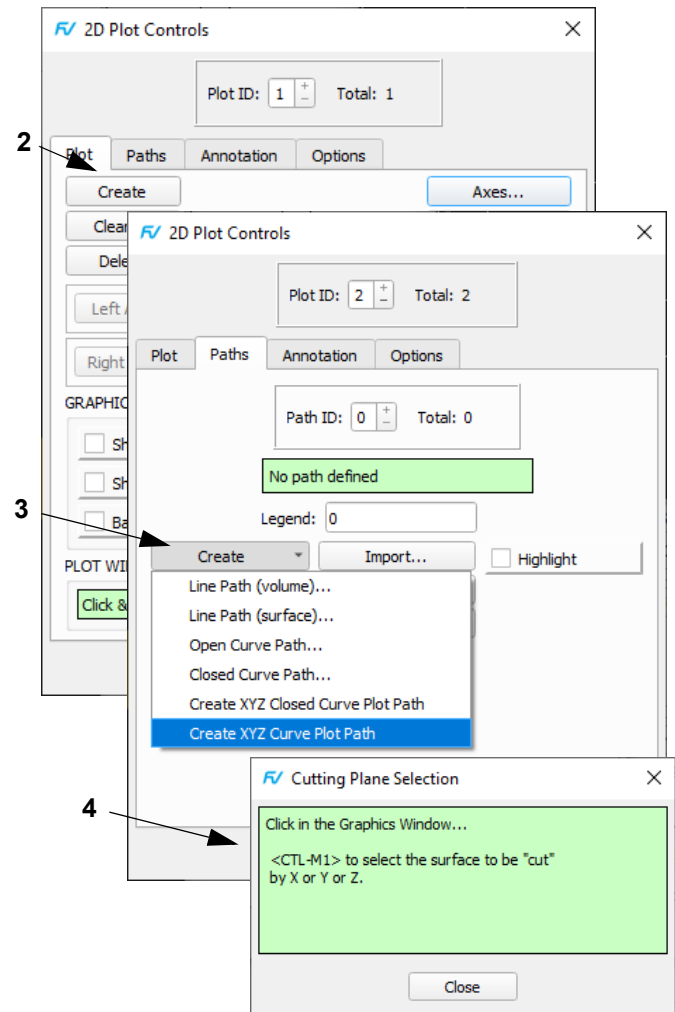


Step 9: Create 2D Plots

We will produce a cut through the tail giving a curve plot showing the airfoil section of the tail along with the value of C_p [PLOT3D] along the path.

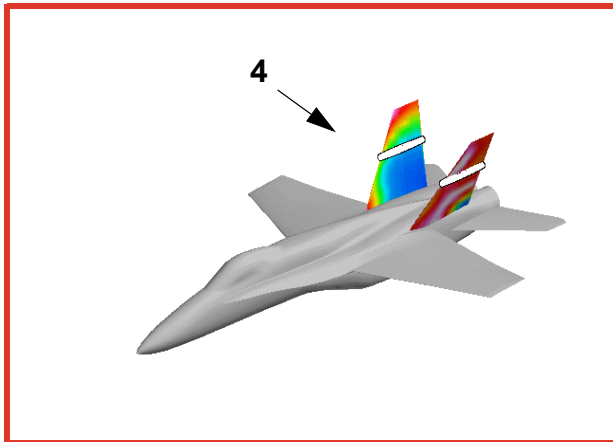


1. Bring up the **2D Plot Controls** panel from either the **Plot** icon on the side toolbar or the **Visualization Panels** pulldown menu. [\[Page 270 Working with FieldView\]](#)
2. Press **Create** on the **Plot** tab of the **2D Plot Controls** panel. This action creates a 2D Plot.
3. Press **Create** on the **Paths** tab of the **2D Plot Controls** panel and choose **Create XYZ Curve Plot Path** from the pulldown menu. This action creates a path within your 2D plot. As a reminder, you can create up to 10 paths within a 2D plot, and up to 9 2D plots.
4. Once you make this selection, you will be prompted with the **Cutting Plane Selection** window to click in the graphics window. Interactively pick the tail surface with the mouse pointer using CTRL-M1 near the center of the vertical stabilizer. This will define the intersection of the cutting plane with the boundary surface.

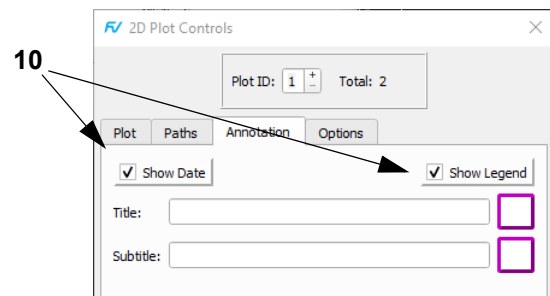
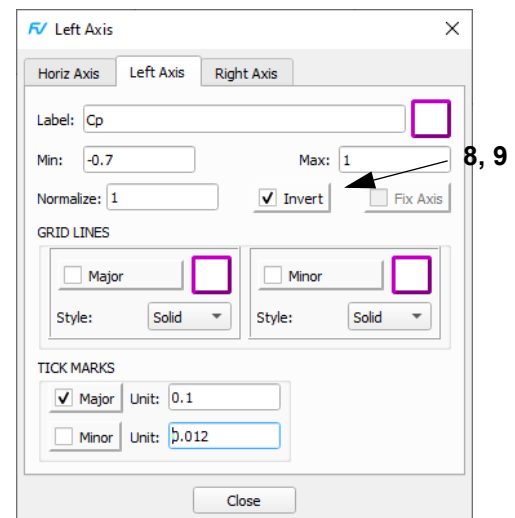
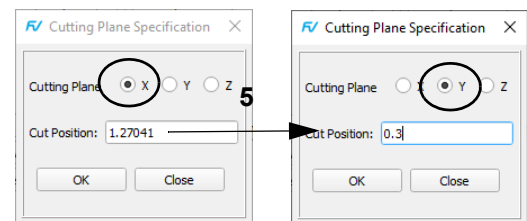
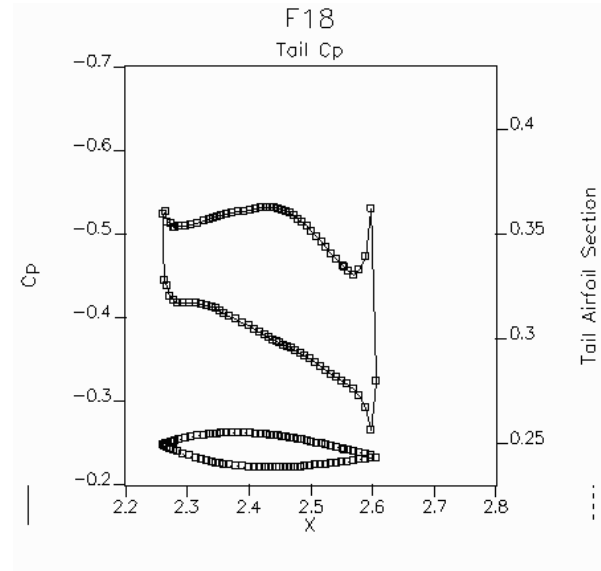


Step 9: Create 2D Plots (cont'd)

Here, we will finalize the appearance of the paths within our 2D plot.

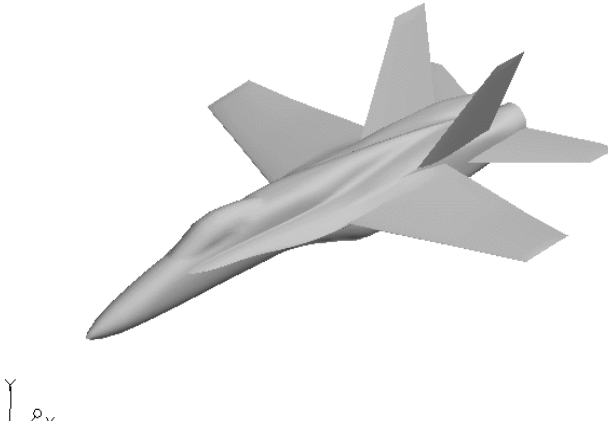


5. Change the cutting plane to **Y**. Change the **Cut Position** to **0.3**. Press **OK**. Then **Close** the **Cutting Plane Specification** panel.
6. On the **Plot** tab of the **2D Plot Controls** panel, change the **Left Axis Function** to **Cp[PLOT3D]**. Turn on the **Show Path** button to better see the planar cut defining the plot path. Also, change the **Path Style** to **Lines & Symbols**. To get the horizontal scale illustrated, press the **Axes...** button; on the **Horiz Axis** tab, ensure **Min**: is 2.2, **Max**: is 2.8, and **TICK MARKS Major** is on with **Unit**: set to 0.1.
7. On the **Plot** tab of the **2D Plot Controls** panel, turn on the **Right Axis** button to display the values of the **Right Axis Function**. Change the **Right Axis Function** to **Z** by pressing the **Select...** button, selecting **Z** from the **Function Selection** panel and pressing **Calculate**.
8. Press the **Right Axis** tab on the **Axis** controls panel. Adjust the **Max**: value of the **Right Axis** to 0.43 to better visualize the thin airfoil section of the tail. Change **TICK MARKS Major Unit**: to 0.05 and **Label**: to **Tail Airfoil Section**.
9. Press the **Left Axis** tab on the **Axis** controls panel. Turn on **Invert** so that the minimum value is at the top of the axis. Change **TICK MARKS Major Unit**: to 0.1. **Close** the **Axis** controls panel.
10. On the **2D Plot Controls** panel, press the **Annotation** tab. Add **Title**: **F18** and **Subtitle**: **Tail Cp**. Turn off the **Show Date** and **Show Legend** buttons, then **Close**.

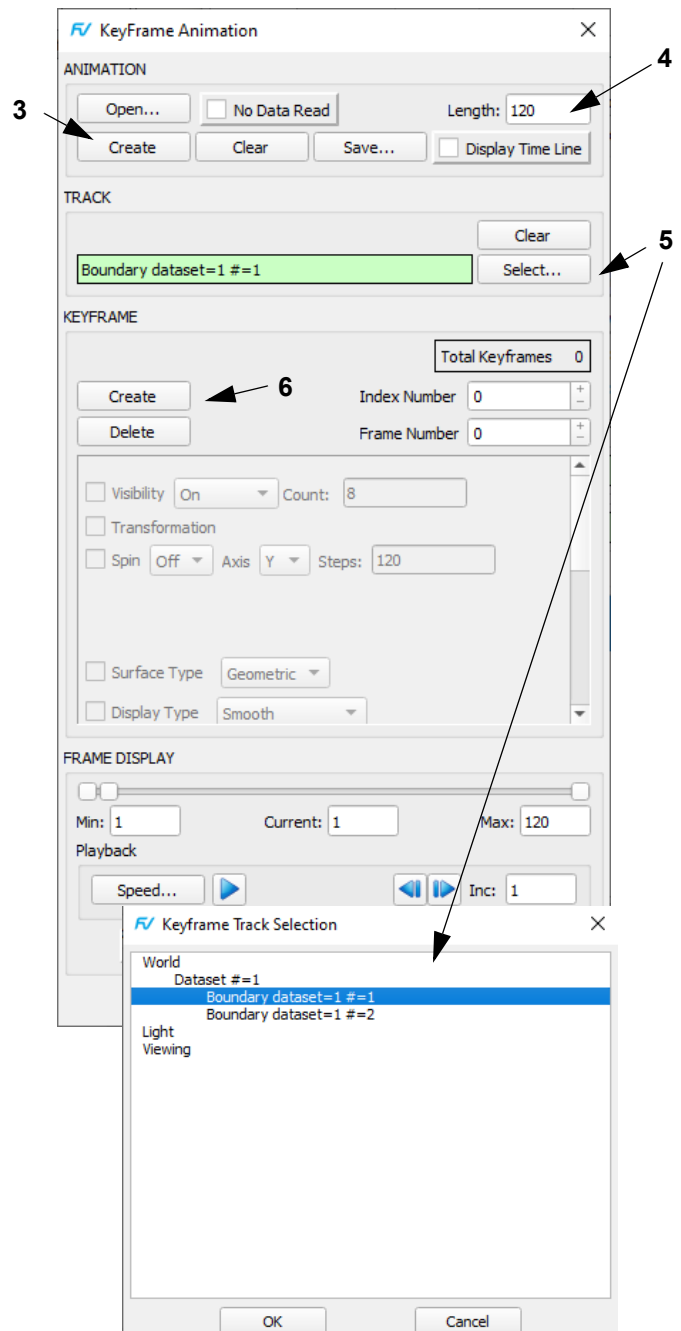


Step 10: Create a Keyframe Animation

This section will show how to create a keyframe animation of 'painting' over a geometric surface while the view is rotating. This section uses a Complete Restart and a Keyframe Restart which accompanies the tutorial data. Keyframe animations require some planning before execution, due to the complexity of the animation that can be created.



1. Copy the data and restarts found in the **/aerospace** sub-directory of the **/tutorial** directory on the **FieldView DVD-ROM** to your local hard-disk. Then read the **Complete Restart** called **f18_keystart.dat** found in the **/aerospace tutorial** directory. This restart creates an image similar to Step 2: Create Boundary Surfaces, where we have read in the data and displayed the body as **Boundary Surfaces**. [\[Page 316 Reference Manual\]](#)
2. Use the **Tools→Graphics Layout Size** pulldown and choose **640x480 NTSC**.
3. Bring up the **Keyframe Animation** panel using the **Tools** pulldown from the Main Menu. Press the **ANIMATION Create** button to create a new keyframe animation.
4. Double-click in the **Length: 120** field and change the default keyframe animation length to **40**, and hit **Enter** or **Return**. The animation will not need to be longer and this will also help display the time line more clearly.
5. Press the **Select...** button. When the **Keyframe Track Selection** panel comes up, click on **Boundary dataset=1 #=1** and press **OK**.
6. Make sure the **Current Frame** in the **Frame Display** section of the panel is 1. Press the **KEYFRAME Create** button to create a keyframe at frame 1 for **Boundary dataset=1 #=1**.



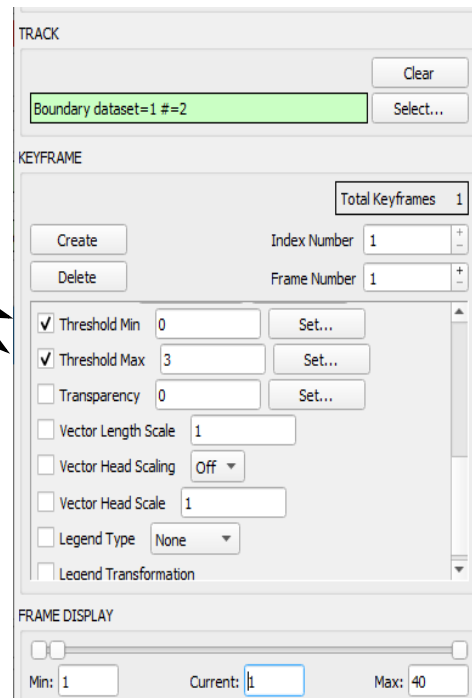
7. Press the **Functions** button on the main toolbar and change the **Threshold** function from **Density (Q1)** to **X**. Alternately, this can be done by changing the **Threshold Function** directly on the **Boundary Surface** panel by pressing the **Select...** button to make the choice from the **Function Selection** panel.
8. Back on the **Keyframe Animation** panel, change the **Threshold Min** to **0** and the **Threshold Max** to **3**.
9. Change the **Current** frame to **40** and press the **KEYFRAME Create** button.
10. Change the **Threshold Min** to **3**. Leave the **Threshold Max** at **3**, but turn on the **Threshold Max** button.

Steps 11 through 15 are similar to those performed on **Boundary dataset=1 #=1**. The corresponding keyframe GUI changes will be similar to those described for steps 5 though 10 and are not explicitly illustrated.

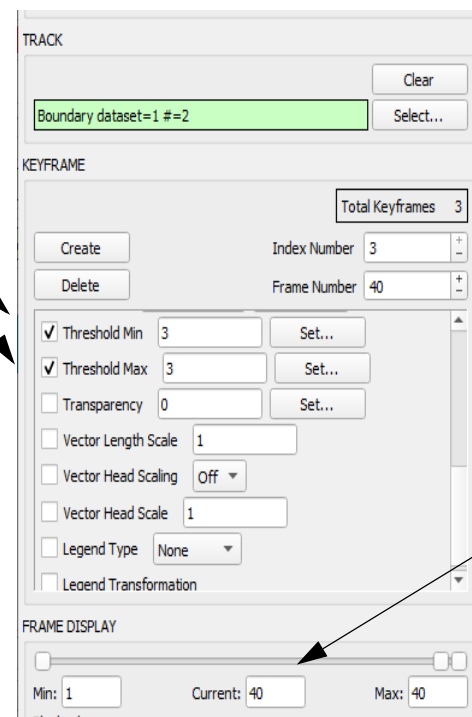
11. Press the **Select...** button and choose **Boundary dataset=1 #=2**. Note that in the **Keyframe Track Selection** panel, **Boundary dataset=1 #=1** now has 2 keys next to it. Press **OK**.
12. Press the **KEYFRAME Create** button to create a keyframe. Note that the **Current Frame** is still **40**.
13. Change the **Threshold Min** to **0**. Leave the **Threshold Max** at **3**, but turn on the **Threshold Max** button.
14. Change the value of **Current** to **1** and press the **KEYFRAME Create** button.
15. Leave the **Threshold Min** at **0**, but turn on the **Threshold Min** button. Change the **Threshold Max** to **0**.
16. Display the **Time Line** for the keyframe animation created so far by turning on **Display Time Line**, then pressing the **OK** button in the **Keyframe Timeline Track Selection** panel. Note how the various keyframes and actions are shown. The **Keyframe Time Line Display** panel can be manually resized. To dismiss the panel, turn off **Display Time Line**.

17. Press the **Play** button to preview your animation, or press the **Frame Advance** arrows to preview specific frames. To speed up the preview, you can change the value of **Inc.** to 2 or 4, thus playing only every other or every 4th frame.
18. Save a keyframe restart by pressing the **Save...** button in the **ANIMATION** section at the top of the **Keyframe Animation** panel.

8



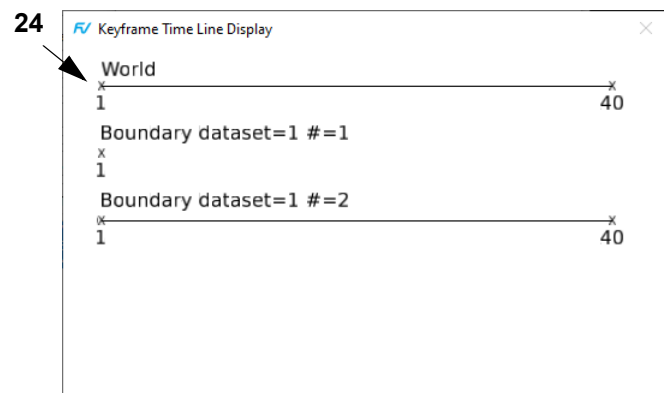
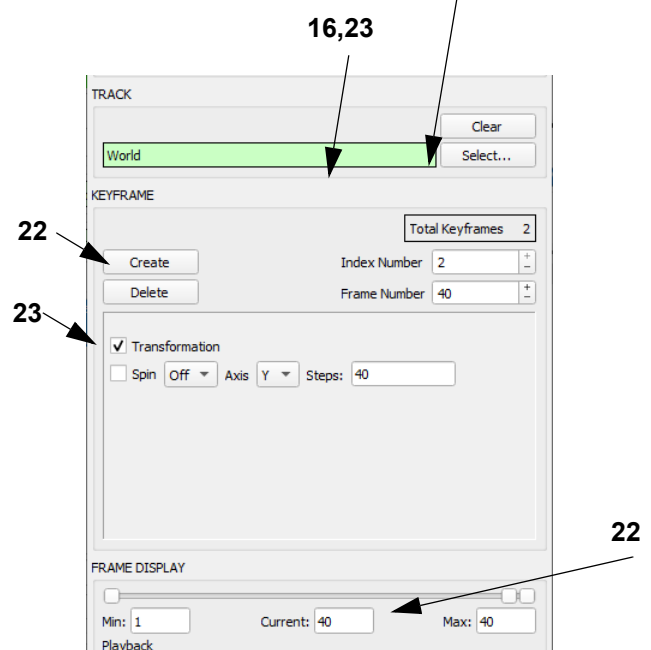
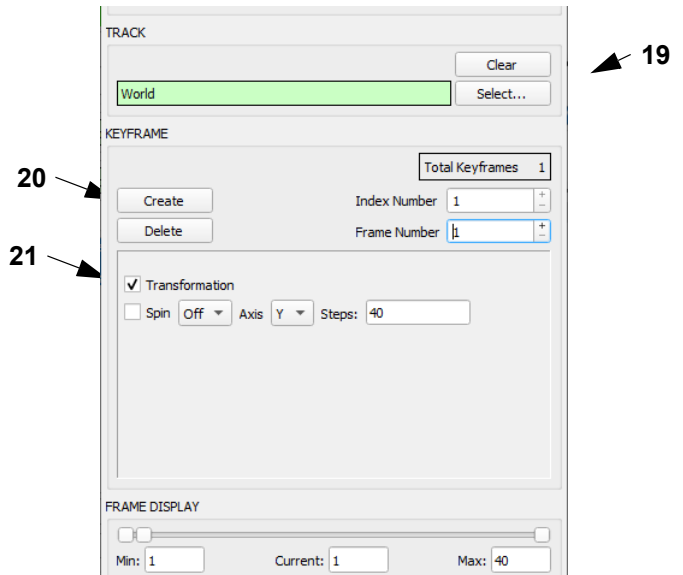
10



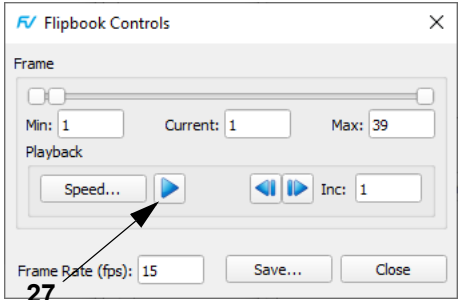
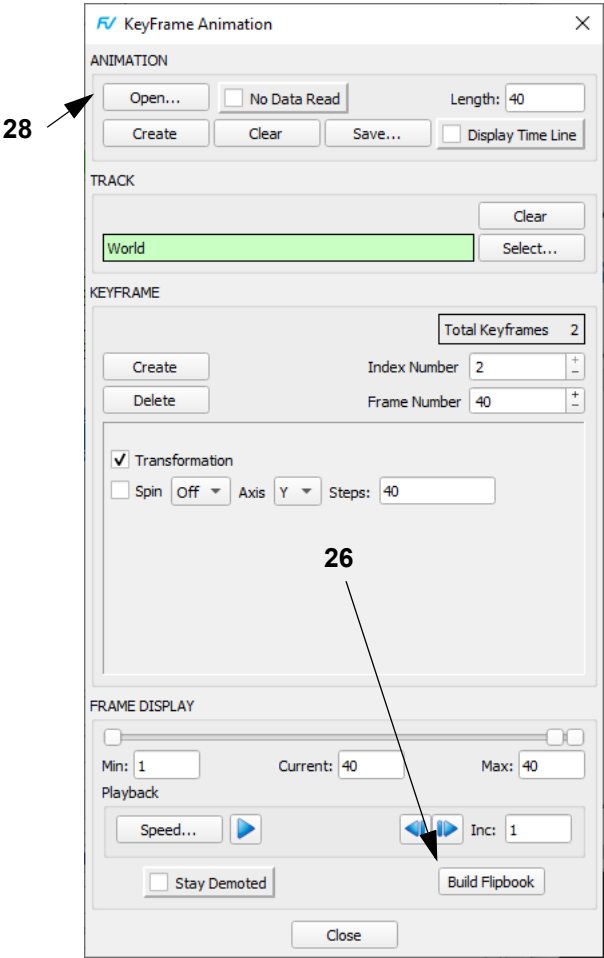
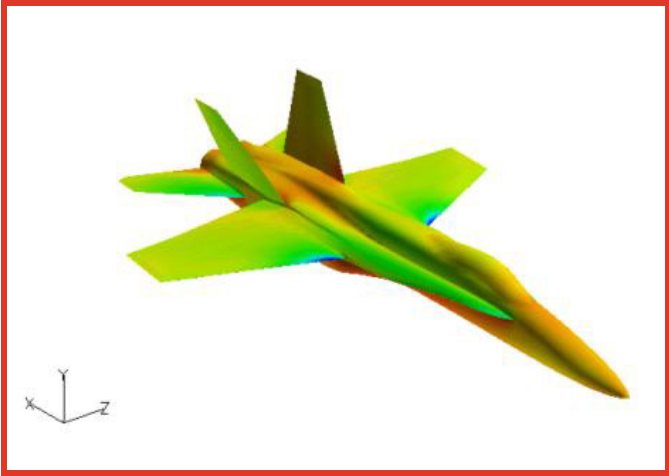
16



19. Finally, we will create the keyframes to perform the rotation of the F18. Change the **Current Frame** to 1. Press the **Select...** button. Choose **World** and press **OK**.
20. Press the **KEYFRAME Create** button to create a keyframe at the **Current Frame** 1.
21. Scrolling to the top of the list if necessary, turn on **Transformation** to set the current view for the first frame.
22. Change the **Current Frame** to 40 and press the **KEYFRAME Create** button.
23. Using the mouse controls on the **Transform Controls** panel, rotate the view of the F18 to some desired position. This rotation will occur during the 'painting' of the body. Ensure that **Transformation** is checked on. This is how you will record the final position of the F18 at frame 40 in your animation.
24. Display the final **Time Line** for the keyframe animation by turning on **Display Time Line**, then pressing the **OK** button in the **Keyframe Timeline Track Selection** panel. Note how the various keyframes and actions are shown.. The **Keyframe Time Line Display** panel can be manually resized. To dismiss the panel, turn off **Display Time Line**.
25. Press the **Play** button to preview your animation, or press the **Frame Advance** arrows to preview specific frames. To speed up the preview, you can change the value of **Inc.** to 2 or 4, thus playing only every other or every 4th frame. Save another keyframe restart.



26. If you have not already changed your Graphics Window to 640x480, do so now. Then press the **Build Flipbook** button. When the **Flipbook Display Size** warning appears, click **OK** to dismiss the warning, build the flipbook and open the **Flipbook Controls** panel.
27. Save the animation by clicking on the **Save...** button in the **Flipbook Controls** panel. [\[Page 309 Reference Manual\]](#)
28. A completed keyframe restart (**f18_anim.key**) has been provided. To load this restart, use the **Open...** button in the **ANIMATION** section of the **Keyframe Animation** panel. This restart will use information stored in the restarts with the rootname, **f18_anim_key.***.



Chapter 4

Free Surface Flow Tutorial



This chapter highlights a Free Surface flow application. It will demonstrate some ways to visualize various types of surfaces found in datasets of this class. In the first part of our visualization, we will read in the data and create a surface of a weir.

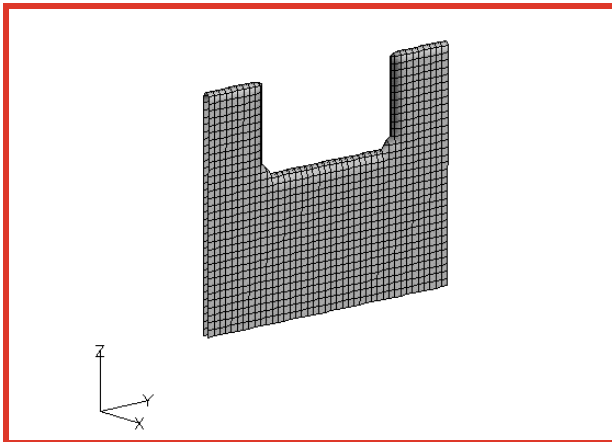
What You Will Learn

- [How to read in FLOW3D® data into FieldView](#)
- [How to visualize a transient dataset](#)
- [How to create an IJK surface with the Computational Surface Panel](#)
- [How to use the Legend](#)
- [How to display velocity vectors on a scalar surface](#)
- [How to change the colormap settings](#)
- [How to display a mesh on a surface](#)
- [How to create a transient flipbook animation](#)
- [How to automatically open a panel by "Quick Picking" its surface](#)
- [How to clip a surface by using Thresholding](#)

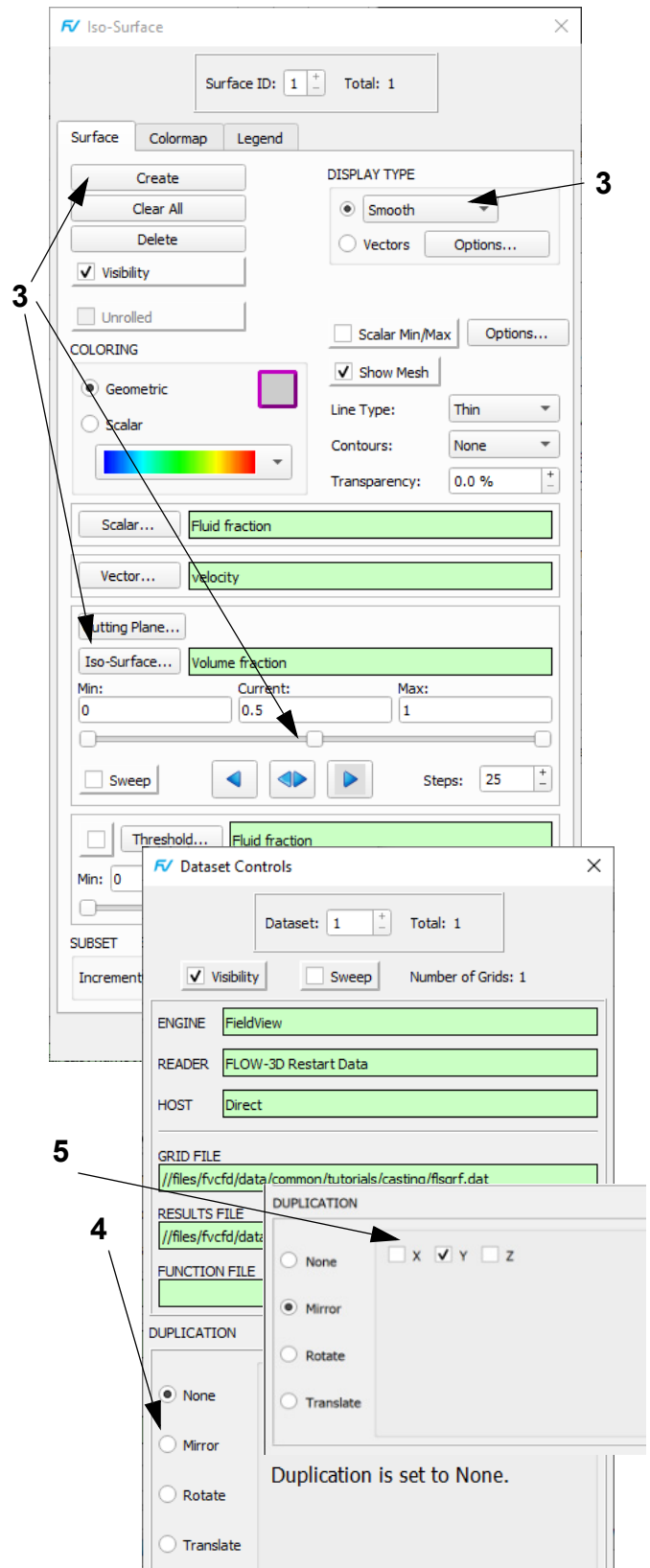
Step 1: Read FLOW-3D® Data



The pictures on each page correspond to the final image you should see when the steps are completed correctly.

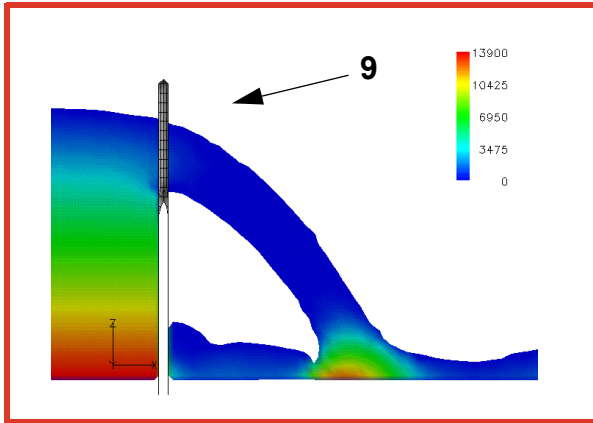


1. Bring up **File** → **Data Input** → **FLOW-3D Restart Data...** Click the **Read Grids & Results** button and read in **flsgrf.dat** from the **casting** tutorial folder. Read in all the Function Names highlighted in the **Function Subset Selection** panel by clicking **OK**. This is a transient dataset and the default will be to read in the last Time Step, 51. Press **OK**, then **Close** the **FLOW-3D® Restart Data** panel. [\[Page 28 Reference Manual\]](#)
2. To visualize the weir, navigate to **Visualization Panels** → **Iso-Surface...** or press the **Iso** icon on the side toolbar. **Create** an Iso-surface.
3. On the **Iso-Surface Create** panel, select **Define Iso Function...** Choose **Volume fraction** and press **Calculate**. [\[Page 170 Working with FieldView\]](#) Set the **Current:** field to **0.5**. Change the **DISPLAY TYPE** from **Constant** to **Smooth** shading and turn on the **Show Mesh** button.
4. Press the **Dataset...** button on the main toolbar to bring up the **Dataset Controls** panel. Press the **Mirror** button in the **DUPLICATION** section.
5. The default **DUPLICATION** mode will be Y. Now your dataset is mirrored about the symmetry plane. **Close** the **Dataset Controls** panels
6. Use **View** → **Outline** on the main menu or the Outline icon on the Transform Controls toolbar to turn the outline off. Use the mouse to orient the weir geometry similar to what is shown above.

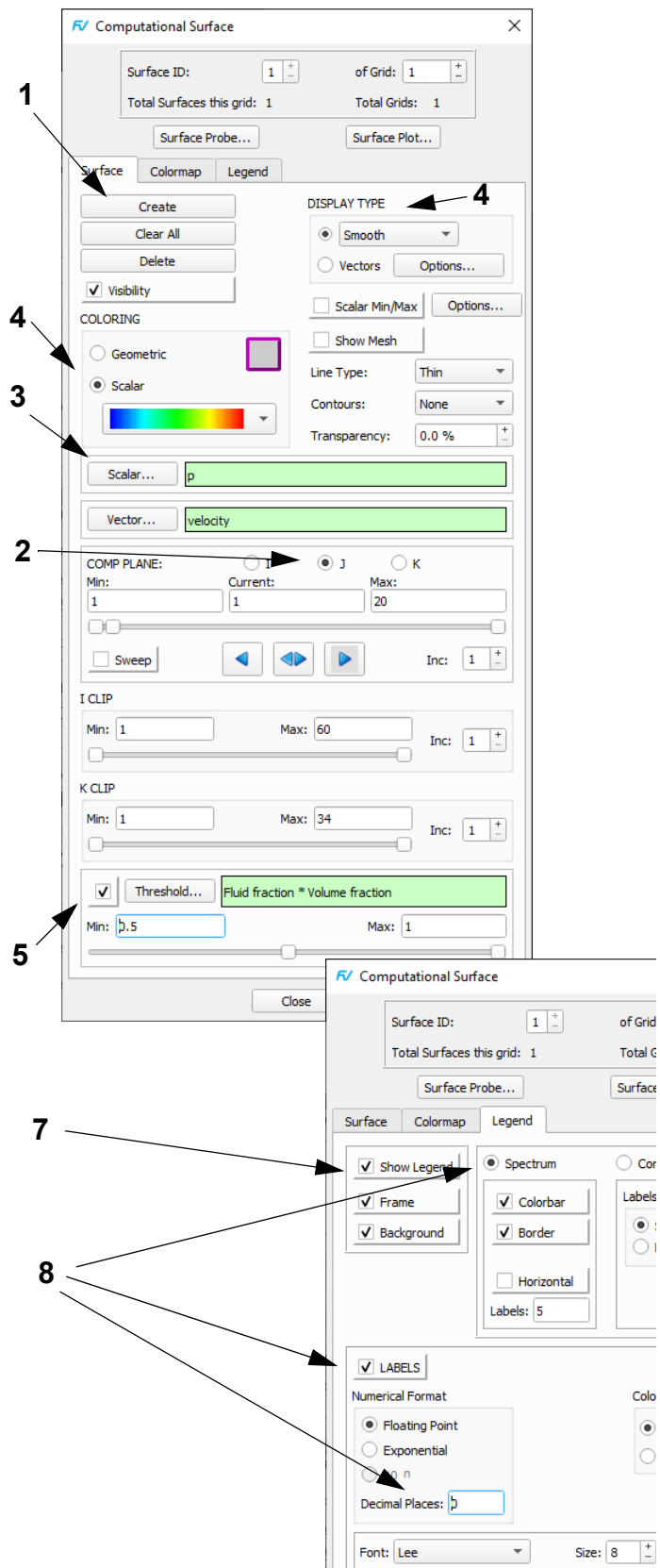


Step 2: Visualize Transient Free Surface Flow

Here, we will visualize the free surface consisting of the fluid passing over the weir.

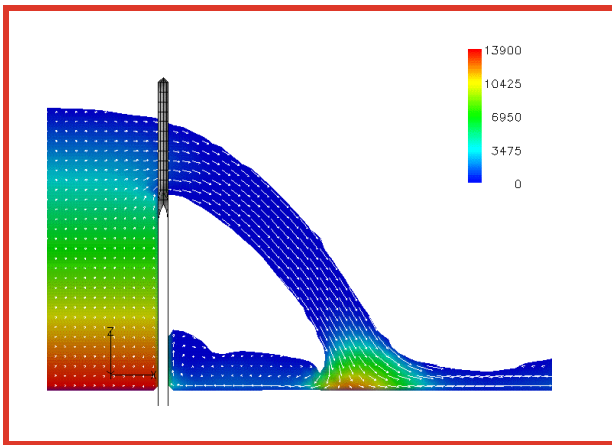


1. To visualize the free surface, navigate to **Visualization Panels** → **Computational Surface...** or press the **Comp** icon on the side toolbar and click **Create**. Change the **COMP PLANE**: from **I** to **J**. [\[Page 161 Working with FieldView\]](#)
2. Move the **J** slider indicating the **Current** value all the way to the left to the **Min**: value.
3. Change the **Scalar Function** to **p** by pressing the **Select...** button, choosing the function from the **Function Selection** panel and pressing **Calculate**.
4. Change **COLORING** to **Scalar**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
5. Change the **Threshold Function** to **Fluid fraction * Volume fraction** and click the **checkbox** to activate. Change threshold **Min**: to **0.5**.
6. From the **Colormap** tab on the **Computational Surface** panel, change the limits to **13900** and **0**.
7. From the **Legend** tab and add a legend by turning on the **Show Legend** option.
8. Ensure the **Spectrum** radio button is on. Change **Labels**: to **5** and **Decimal Places**: to **0**. Use **SHIFT+Left** mouse to reposition the legend as desired.
9. Orient the dataset using **View** → **Defined Views...** or the button on the **Transform Controls** toolbar. Choose **-Y** and **Close** the panel. Use the mouse to create a view similar to the one illustrated above.

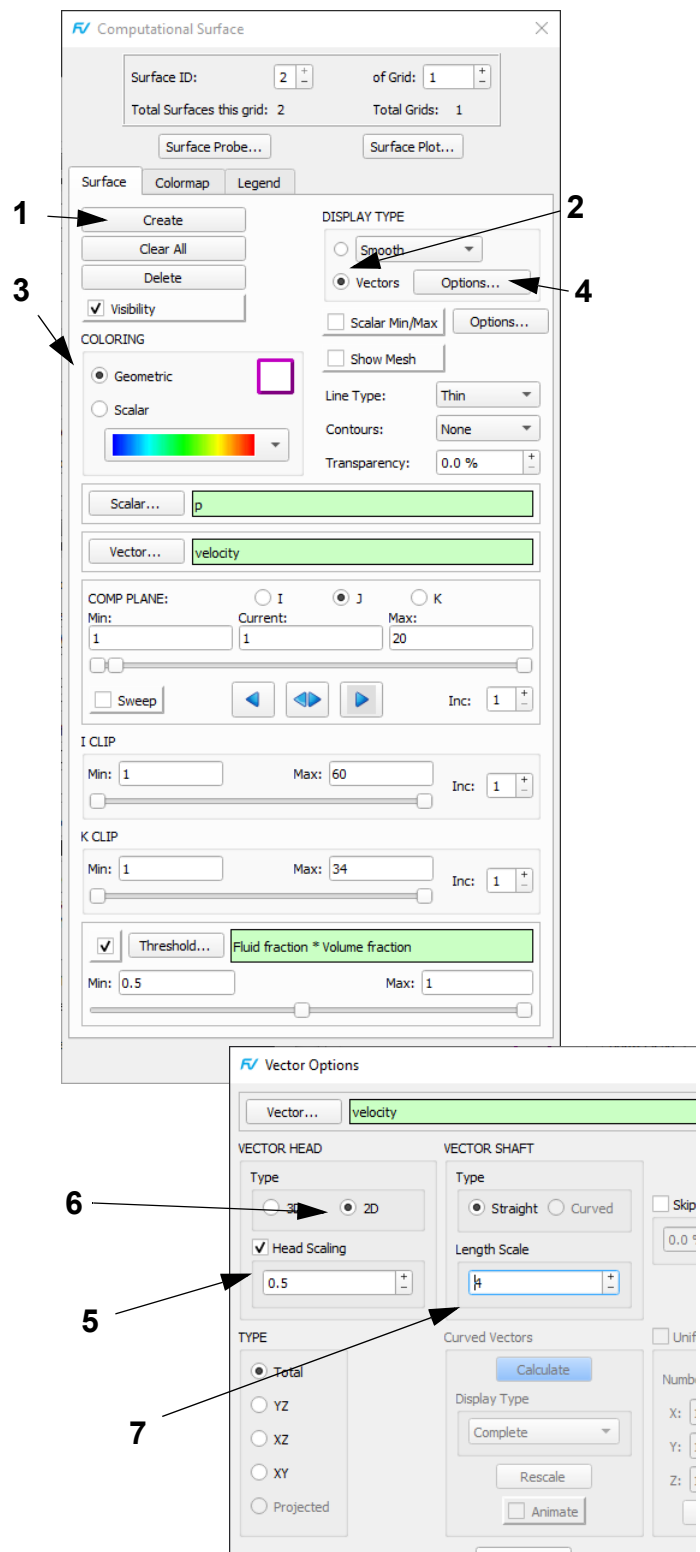


Step 3: Display Velocity Vectors on a Surface

In this step, we will place vectors on the free surface consisting of the fluid passing over the weir.

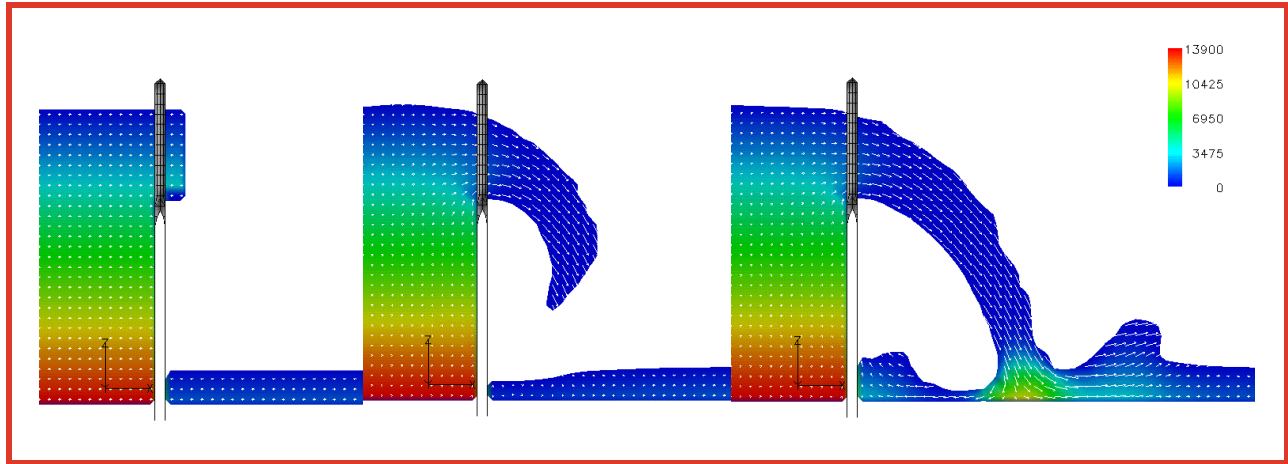


1. Switch back to the **Surface** tab on the **Computational Surface** panel and press **Create** to create a second computational surface. This one will inherit several of the attributes of the first surface.
2. Change the **DISPLAY TYPE** to **Vectors**.
3. Change **COLORING** from **Scalar** to **Geometric**. Adjust the **Geometric** color for the vectors on the **Colormap Specification** panel as desired. (They are shown white above.)
4. Press the **Options...** button to the right of the **Vectors** radio button to change the display attributes of the vectors.
5. On the **Vector Options** panel, turn on **Head Scaling** and change the value to **0.5**.
6. Note that the default vector head **Type** is 2D.
7. Change the **Length Scale** to **4**. Close the **Vector Options** panel.

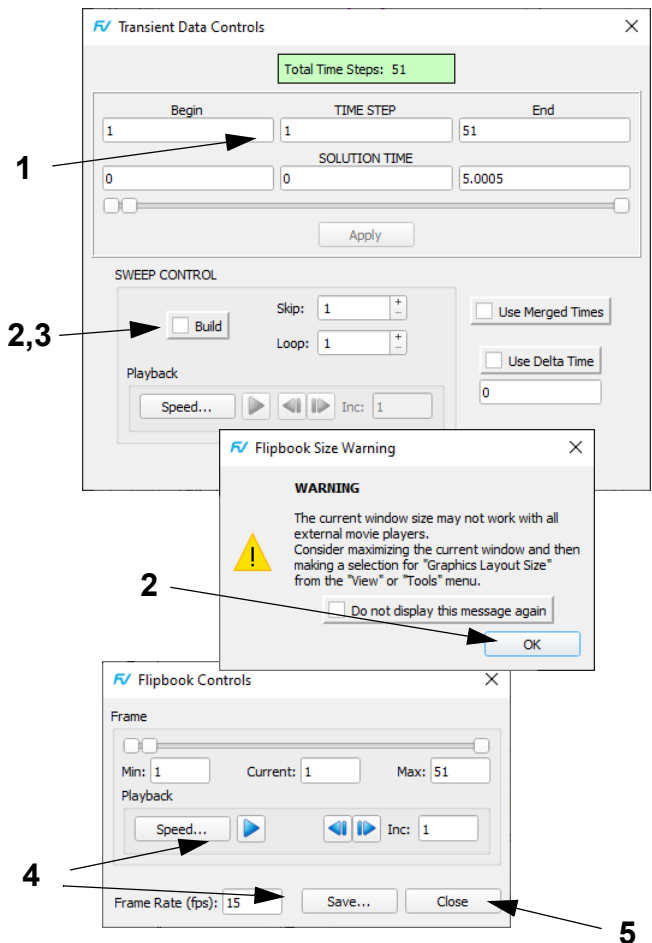


Step 4: Create a Transient Flipbook Animation

In this step, we will create a flipbook or animated sequence consisting of the entire transient data-set.

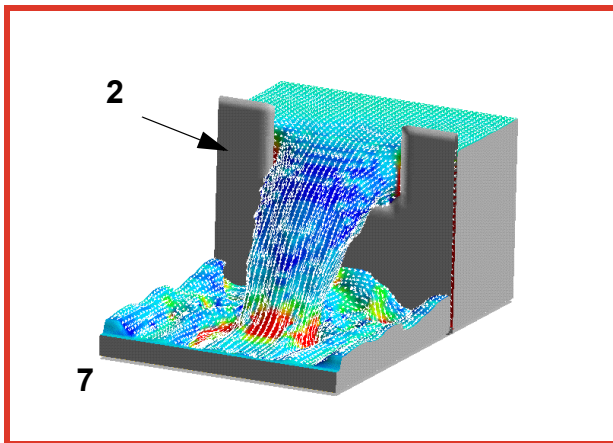


1. Bring up the **Transient Data Control** panel from the **Tools** pulldown menu. Change the current **TIME STEP** to **1** and press the **Apply** button. This will load the first time step. The image in the graphics window will change to the image shown in the upper left (above). The images above represent **TIME STEP** values of **1, 2, and 3**
2. Navigate to **Tools** → **Flipbook Build Mode**. When the **Flipbook Display Size Warning** appears, click **OK** to dismiss it. This will change the button from **Sweep** to **Build** on the **Transient Data Controls** panel.
3. Press the **Build** button to start creating the flipbook. You can stop the flipbook build by clicking the **Stop** button on the **Building Flipbook** panel (not shown).
4. Once the build is complete, the graphics window becomes a playback window. The **Flipbook Controls** panel can be used to **Play** or **Save** the flipbook. **Play** the flipbook and then **Save** it. (To return to an interactive **FieldView** graphics window, you must **Close** the **Flipbook Controls** panel.)
5. **Close** the **Flipbook Control** panel.

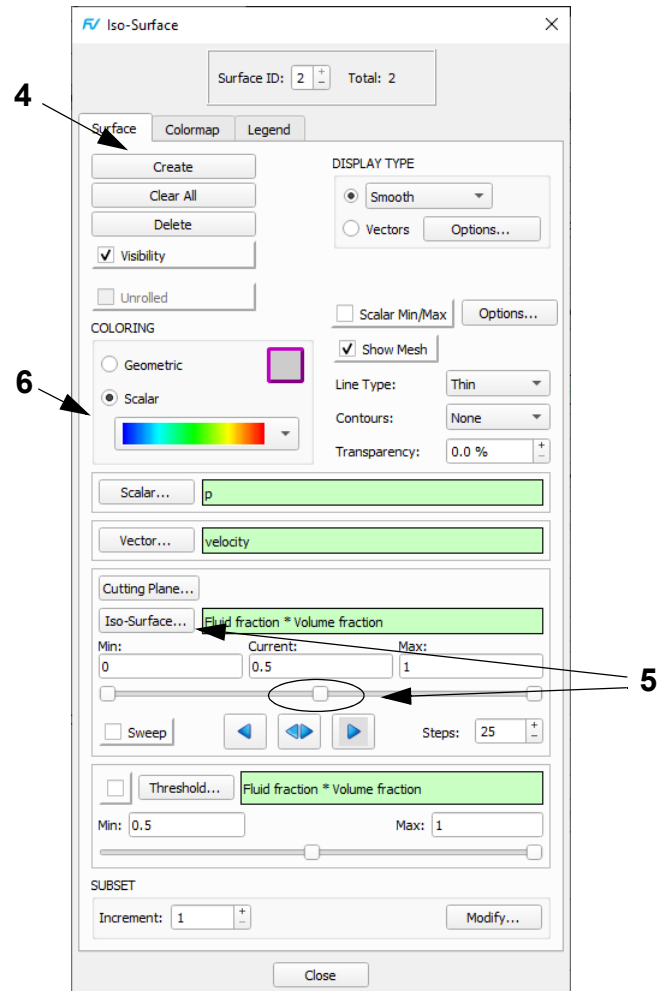


Step 5: Create an Advanced Visualization

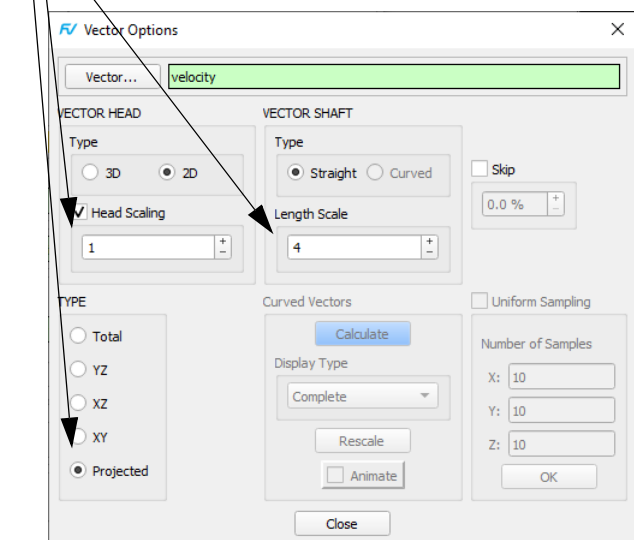
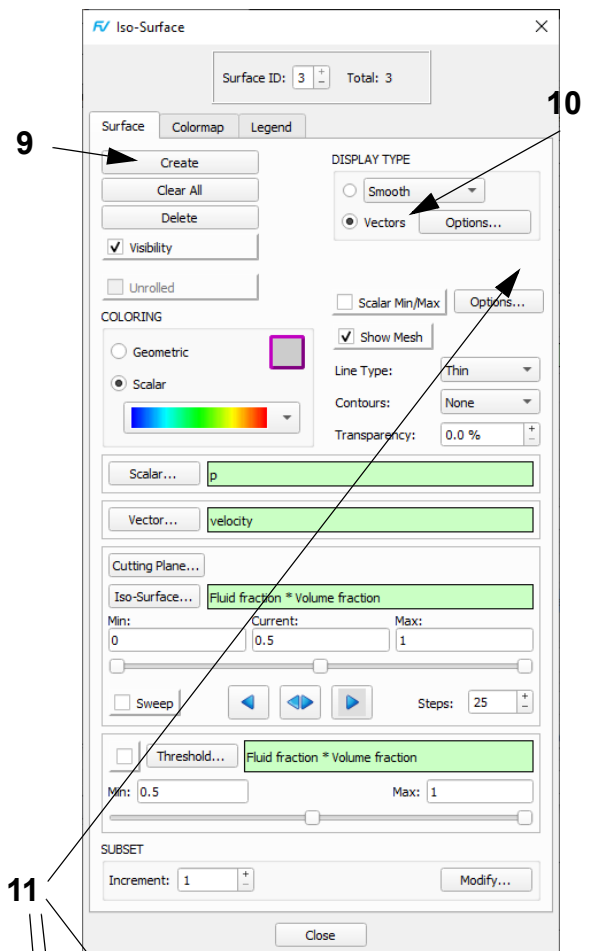
In this step, we look at a more advanced method of visualizing the free surface.



1. Turn off the **Visibility** of the two computational surfaces we have used in the preceding steps.
2. Quick-pick (double-click) on the weir iso-surface. This will bring up the **Iso-Surface** panel. Turn off the **Show Mesh** button.
3. On the **Transient Data Controls** panel, set the current **TIME STEP** to **14** and press Enter followed by the **Apply** button, then **Close**.
4. Press **Create** to create a second iso-surface.
5. Click **Iso-Surface...** for the **Iso Function** and choose **Fluid fraction * Volume fraction** from the **Function Selection** panel. Press **Calculate**. Set the **Current** value of the **Iso function** to **0.5**.
6. Change the **COLORING** from **Geometric** to **Scalar**.
7. Set the minimum and maximum (range) on the **Color-map** tab to **-250** and **950** respectively (not shown).
8. Rotate and resize the data with the mouse to approximate the view seen above.



9. **Create** a third **Iso-surface**.
10. Change the **DISPLAY TYPE** from **Smooth** to **Vectors**.
11. Press the **Vectors Options...** button and turn on **Head Scaling** with a value of **1**, set the **Length Scale** to **4** and change **TYPE** to **Projected**. **Close** the **Vector Options** panel.
12. To get the front and sides of the fluid boundary we need to create three thresholded computational surfaces. Switch back to the **Computational Surface** panel. Create the following additional computational surfaces. For each, set **COLORING** to **Scalar** and **DISPLAY TYPE** to **Constant** shading:
 13. For the front, **Create** a **COMP PLANE** for **I**, setting **I = 60**. Set the **Threshold Function** to **Fluid fraction * Volume fraction**, **Min:** to **0.5**, and turn **Threshold Clip** on.
 14. For the sides, **Create** a **COMP PLANE** for **J**, setting **J = 1** for one side and **J = 20** for the other. Similar to the other computational surface, set the **Threshold Function** to **Fluid fraction * Volume fraction**, **Min:** to **0.5**, and turn **Threshold Clip** on.
15. Bring up the **Transient Data Controls** panel from the **Tools** pulldown menu. Change the current **TIME STEP** to **1** and press the **Apply** button to load the first time step.
16. Navigate to **Tools** → **Flipbook Build Mode**. When the **Flipbook Display Size Warning** appears, click **OK** to dismiss it. This will change the button from **Sweep** to **Build** on the **Transient Data Controls** panel.
17. Press the **Build** button to start creating the flipbook. You can stop the flipbook build by clicking the **Stop** button on the **Building Flipbook** panel (not shown).
18. Once the build is complete, the graphics window becomes a playback window. The **Flipbook Controls** panel can be used to **Play** or **Save** the flipbook. **Play** the flipbook and then **Save** it. (To return to an interactive **FieldView** graphics window, you must **Close** the **Flipbook Controls** panel.)



Chapter 5

Basic

Automotive

Tutorial

5

This chapter will highlight an application from the automotive industry. It will demonstrate some ways to visualize various types of surfaces found in datasets of this class.

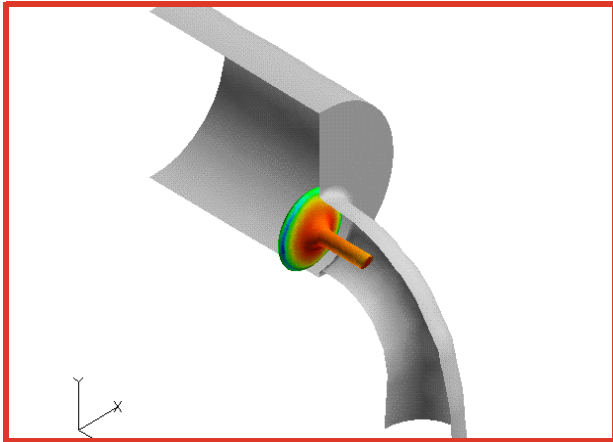
What You Will Learn

- [How to read in a FLUENT-UNS file](#)
- [How to cut away a portion of a surface](#)
- [How to visualize vortex formation using velocity vectors](#)
- [How to change velocity vector settings](#)
- [How to visualize vortex formation using streamlines](#)
- [How to automatically seed a surface](#)
- [How to display shaded streamlines](#)
- [How to create a flipbook animation and save it to a file](#)
- [How to create a 2D Plot of shear reate and density on a valve](#)
- [How to integrate a scalar on a surface](#)

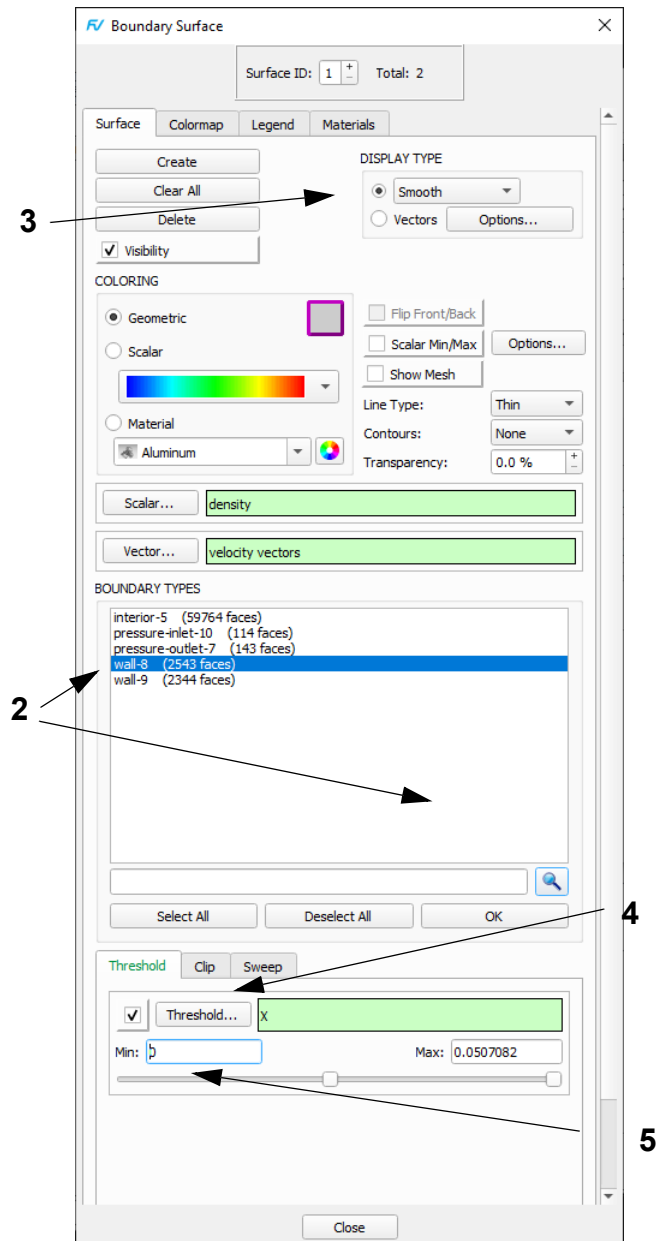
Step 1: Read in FLUENT-UNS Data



The pictures on each page correspond to the final image that should be created when the steps are completed correctly.

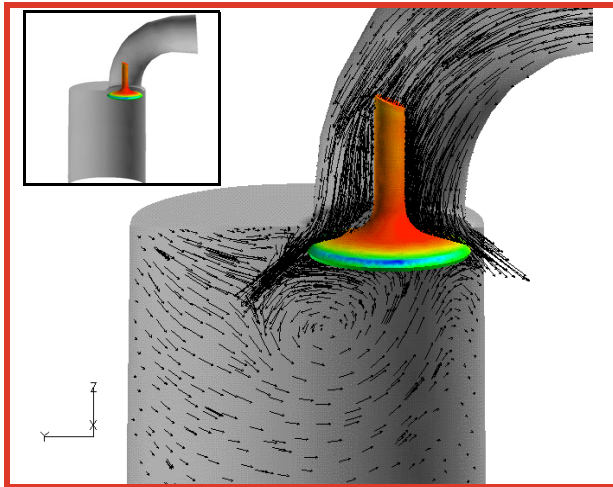


1. Bring up **File** → **FLUENT [FV-UNS Export]**. Read **Grid or Combined Data intake.uns**. Close the **FLUENT [FV-UNS Export]** panel. [\[Page 85 Reference Manual\]](#)
2. Click the **Bound** icon on the side toolbar. **Create** the first of two Boundary Surfaces and select the **BOUNDARY TYPE** named **wall-8**. Press **OK**. [\[Page 240 Working with FieldView\]](#)
3. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
4. Press the **Threshold** button and set the **Threshold Function** to **X** on the **Function Selection** panel [\[Page 100 Reference Manual\]](#), and activate the function by clicking the checkbox.
5. Set the **Threshold Clip Min**: field to **0**. This will threshold away the left side of the intake valve housing, exposing the valve itself and the internal flow.
6. **Create** a second Boundary Surface (panel not shown), select **wall-9** and press **OK**. This surface will inherit the attributes of the first, i.e. it will be smooth shaded, geometric colored and thresholded. Change the **COLORING** from **Geometric** to **Scalar** (density). Turn **Threshold Clip** off to make the entire valve visible.
7. From the **View** pulldown menu, turn off **Outline** and **Perspective**. [\[Page 52 Working with FieldView\]](#)

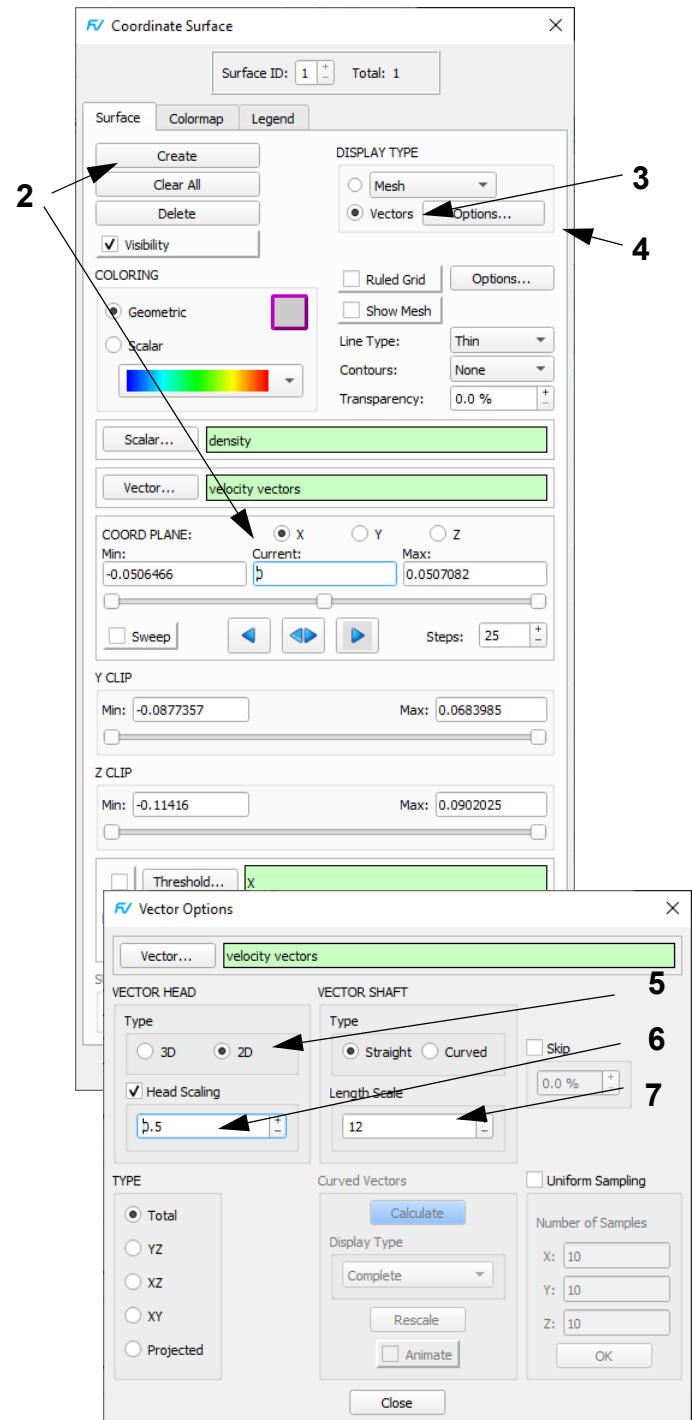


Step 2: Capture Vortex Formation with Velocity Vectors

In this step we will reorient the model and create a coordinate surface and display it as vectors to capture the recirculating flow around the valve.

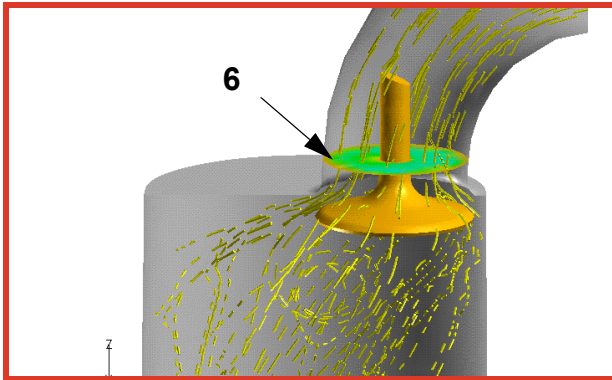


1. First we will orient our model. Press the icon on the **Transform Controls** toolbar to bring up the **Defined Views** panel. Press the **-X VIEWING DIRECTION** and **Close** the panel. Click the left-most button on the **Viewer Toolbar** to toggle to locked transform mode. [\[Page 30 Working with FieldView\]](#) Use the Z rotation button pair with the right mouse button to turn the dataset approximately 90 degrees, followed by scale and X rotation, to match the illustration above.
2. **Create** an **X** Coordinate Surface. Set the **Current** value for X to 0. You can do this by either typing in the value or dragging the slider. [\[Page 232 Working with FieldView\]](#)
3. Change the **DISPLAY TYPE** from **Mesh** to **Vectors**.
4. Bring up the **Vector Options** panel.
5. Note that the default **VECTOR HEAD** Type is **2D**.
6. Turn on **Head Scaling**. Change the **Head Scaling** value to **0.5**.
7. Change the **Length Scale** to **12** and **Close** the **Vector Options** panel.
8. Press the color chip next to the **Geometric** selector. On the **Geometry Color** panel, choose the black chip and **Close**.

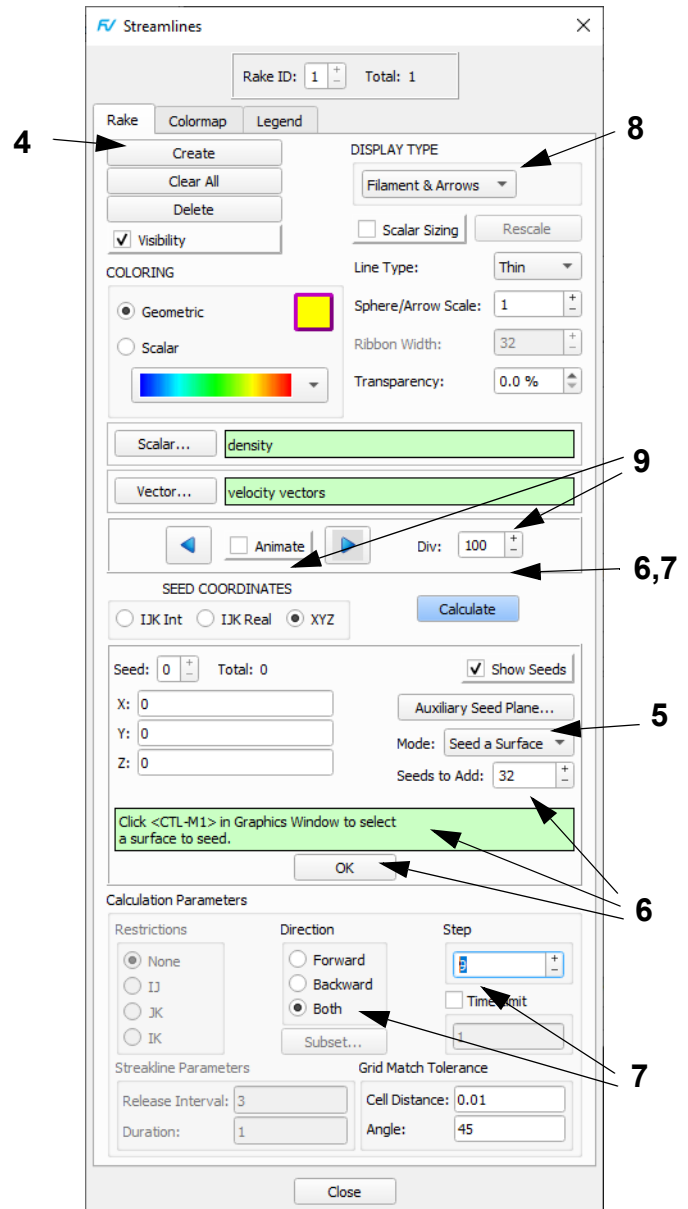


Step 3: Display Vortex Formation with Streamlines

In this step we will create another coordinate surface and use it as a streamline seeding plane. We will compute streamlines to show the recirculating flow around the valve.

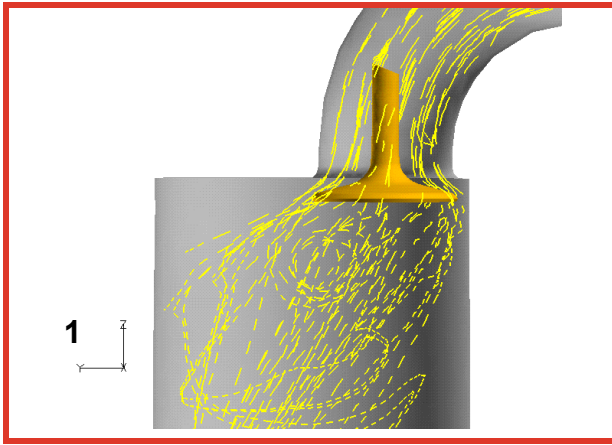


1. First turn off the **Visibility** of the vector Coordinate Surface (not shown).
2. **Create** a second Coordinate Surface. Turn the **Visibility** on, change the **COLORING** to **Scalar** and the **DISPLAY TYPE** to **Smooth** shading. Change the **COORD PLANE** to be a **Z=0.01** surface. We will use this surface to seed on for our streamline rake.
3. Press the color chip next to the **Geometric** selector. On the **Geometry Color** panel, choose the yellow chip and **Close**.
4. Bring up the **Streamlines** panel and press **Create** on the **Rake** tab. [\[Page 184 Working with FieldView\]](#)
5. Change the seeding **Mode:** from **Add** to **Seed a Surface**. Pick the **Z** Coordinate surface by pointing to the surface and pressing **CTRL-M1** on the surface in the graphics window.
6. You should see an "x" on the surface. The message in the **Streamlines** panel should now read **Adjust "Seeds to add:" value; then press OK**. Change the **Seeds to add:** to **32** press the **OK** button to distribute 32 seeds on the surface. Press **Calculate**.
7. In the **Calculation Parameters** section of the **Streamlines** panel, change the **Direction** to **Both** and **Step** to **9**. Press the **Calculate** button again.
8. Change the **DISPLAY TYPE** to **Filament & Arrows**.
9. Turn off **Show Seeds**. Change **Div:** to **100**. Turn on **Animate**. Turn **Animate** off for the next step.

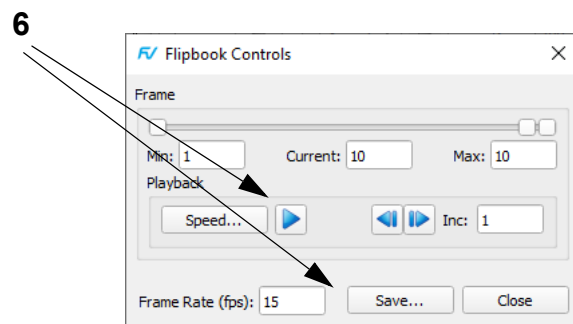
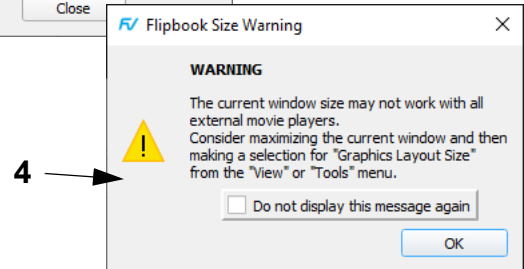
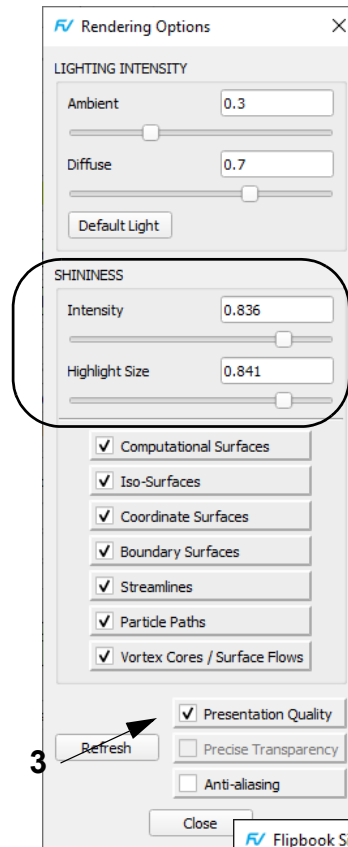


Step 4: Create a Flipbook Animation

In this step we will create a flipbook of the animated streamlines. The flipbook will then be saved for later playback.

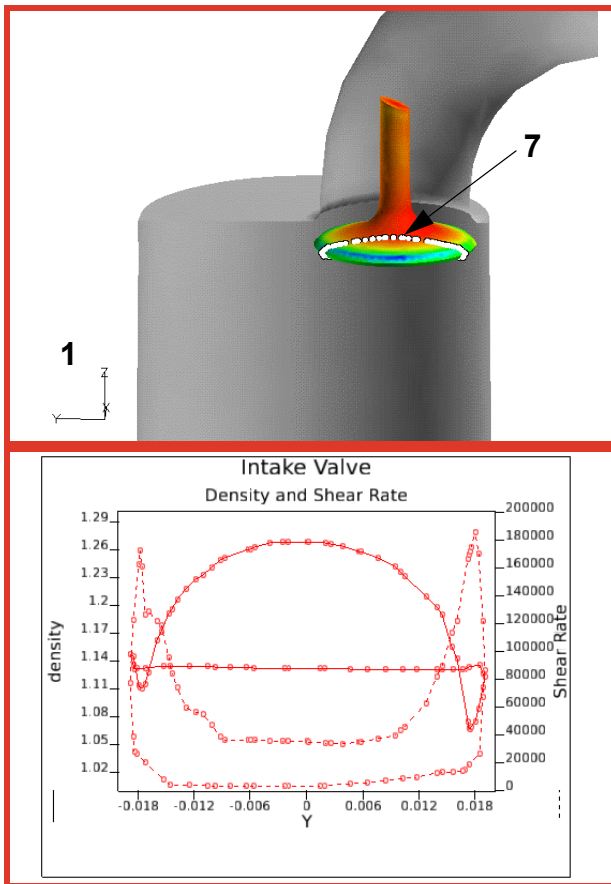


1. Realign the intake valve assembly by toggling to Multi-Transform mode and using M2 to rotate the assembly up slightly. Roughly align it so that the X axis points out of the screen (step not shown).
2. Change the **Line Type** of the streamlines to **Medium**.
3. From the Main Menu, go to **View** → **Rendering Options**. Turn on **Presentation Render** and adjust the **Intensity** and **Highlight Size**. Press the **Refresh** button to update the graphics window. **Close** the panel. [\[Page 58 Working with FieldView\]](#)
4. Navigating to **Tools** → **Flipbook Build Mode**. When the **Flipbook Display Size Warning** appears, click **OK** to dismiss it. This will change the button from **Animate** to **Build** on the **Streamlines** panel.
5. Press the **Build** button on the **Streamlines** panel to create a 10 frame flipbook. Wait until the build is finished. [\[Page 309 Reference Manual\]](#)
6. Once the build is complete, the graphics window becomes a playback window. The **Flipbook Controls** panel can be used to **Play** or **Save** the flipbook. **Play** the flipbook and then **Save** it. (To return to an interactive **FieldView** graphics window, you must **Close** the **Flipbook Controls** panel.)

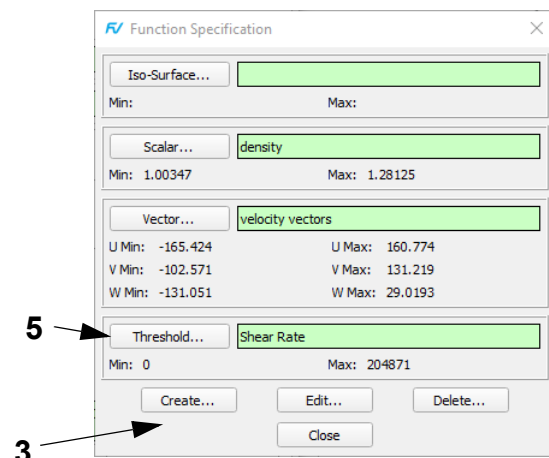
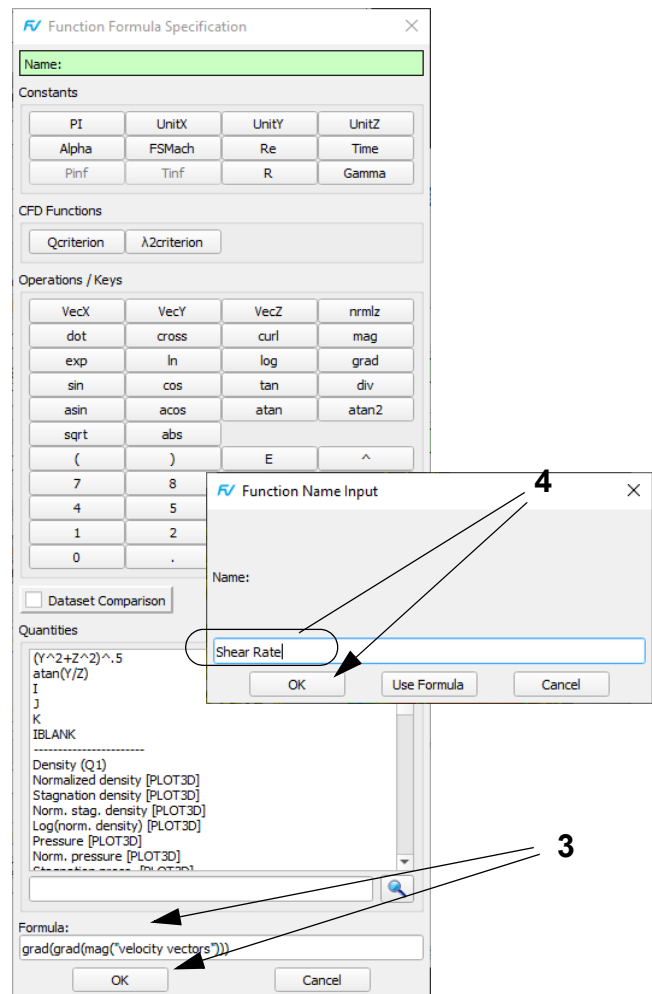


Step 5: Create a 2D Plot Along a Surface

In this step we will create a new function and plot this function along a cut of the intake valve.

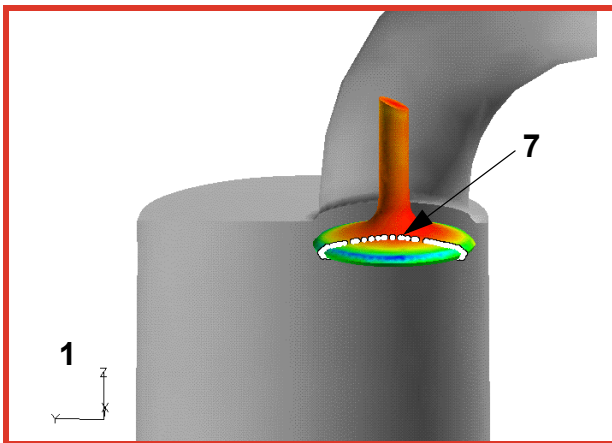


1. Rotate the intake valve assembly down by using M2. Align it so that the valve can be better seen.
2. Turn off the **Visibility** of the streamlines and the Z coordinate surface.
3. Press the **Functions** button on the main toolbar to bring up the **Function Specification** panel. Press **Create** to launch the **Function Formula Specification** panel. Here, create a function defined as $\text{mag}(\text{grad}(\text{mag}(\text{"velocity vectors"})))$. Press **OK** when done. [\[Page 100 Reference Manual\]](#)
4. Type in **Shear Rate** in the resulting **Function Name Input** panel and press **OK**.
5. Press the **Threshold...** button on the **Function Specification** panel, and choose the new function **Shear Rate** from the Function Selection panel. Press **Calculate** to load it into the Threshold register. **Close** the panel.

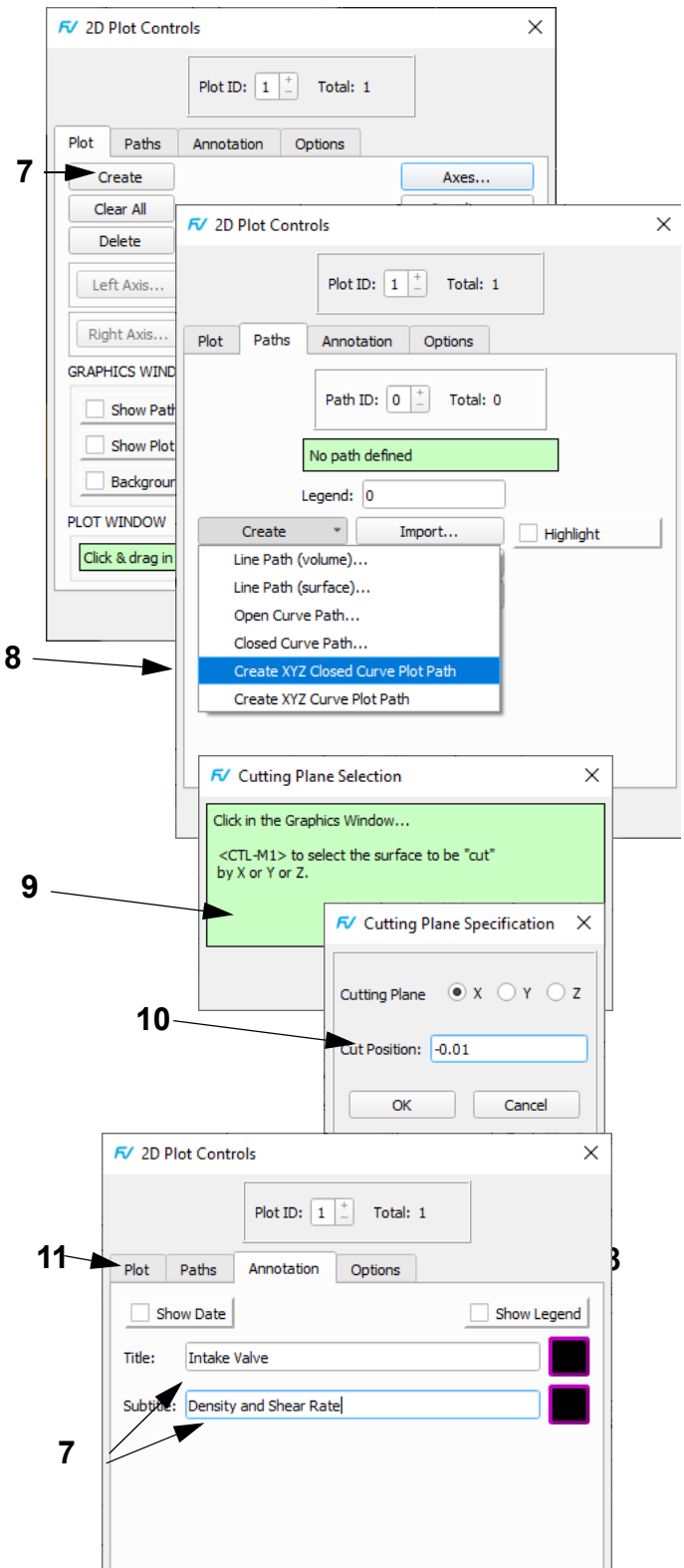


Step 5: Create a 2D Plot Along a Surface (cont'd)

Here, we complete the creation of the 2D plot.

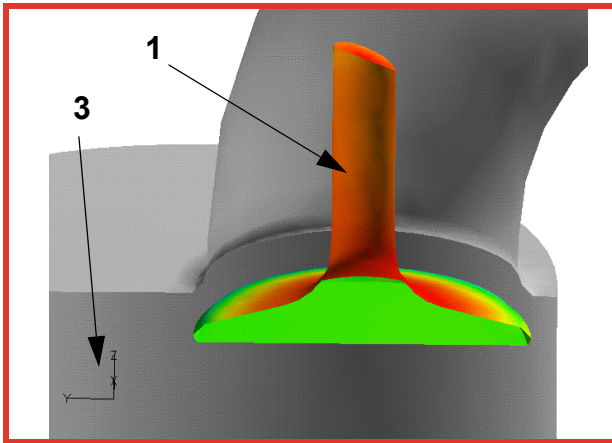


6. Bring up the **2D Plot Controls** panel from either the **Plot** icon on the side toolbar or the **Visualization Panels** pulldown menu. [\[Page 270 Working with Field-View\]](#)
7. Press **Create** on the **Plot** tab of the **2D Plot Controls** panel. This action creates a 2D Plot.
8. Press **Create** on the **Paths** tab of the **2D Plot Controls** panel and choose **Create XYZ Closed Curve Plot Path** from the pulldown menu. This action creates a path within your 2D plot. As a reminder, you can create up to 10 paths within a 2D plot, and up to 9 2D plots.
9. After making this selection, the **Cutting Plane Selection** panel will prompt you to click in the graphics window. Interactively pick on the valve surface (as shown above) with the mouse pointer using **CTRL-M1**. This will define the intersection of the cutting plane and the boundary surface.
10. On the **Cutting Plane Specification** panel, change the Cut Position to -0.01 and press **OK**. **Close** the **Cutting Plane Specification** panel.
11. On the **Plot** tab of the **2D Plot Controls** panel, turn on **Show Path** and **Right Axis**, and change the **Horizontal Axis/Plotting Direction** to **Y**. Ensure that the Left Axis and Right Axis Functions are **density** and **Shear Rate**. Change the **Path Style** to **Lines & Symbols**.
12. Press the **Annotation** tab, add the titles shown, and turn off **Show Date** and **Show Legend**.

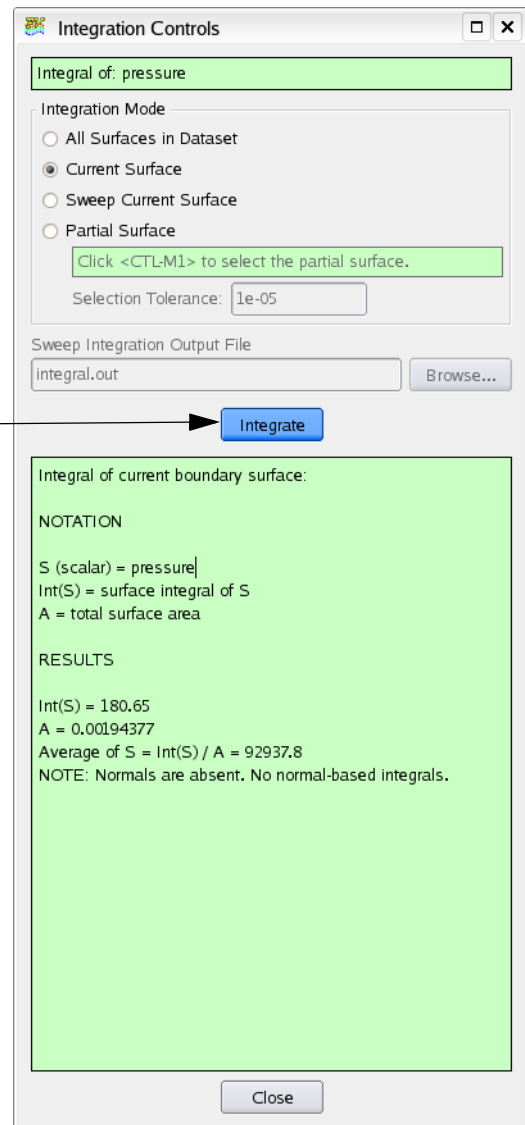


Step 6: Integrate Pressure

In this step we will integrate pressure over the intake valve.



1. Quick-pick (double-click) the intake valve. This will make the boundary surface (valve) the current surface and bring up the **Boundary Surface** panel. All actions take place on the current surface. [\[Page 75 User's Guide\]](#)
2. On the **Boundary Surface** panel, click on the **Select...** button for the **Scalar Function**, change to pressure on the **Function Selection** panel and **Calculate**. Also, change the **Threshold Function** to X. At this point, the **Threshold Clip** button should be off.
3. Use M3 to zoom into the valve.
4. Navigate to **Tools** → **Integration...** to bring up the **Integration Controls** panel. Change the **Integration Mode** to **Current Surface**. Press the **Integrate** button. The result (displayed on the **Integration Controls** panel) gives the integration values for the entire valve surface.
5. Turn on **Threshold Clip** for the valve on the **Boundary Surface** panel and enter **0** for **Min:** (action not shown). This should display only one half of the valve, as seen above.
6. Back on the **Integration Controls** panel, press **Integrate** again. The result gives the integrated values for only one half of the valve surface. **Close** the **Integration Controls** panel.



Chapter 6

Basic Combustion Tutorial



What You Will Learn

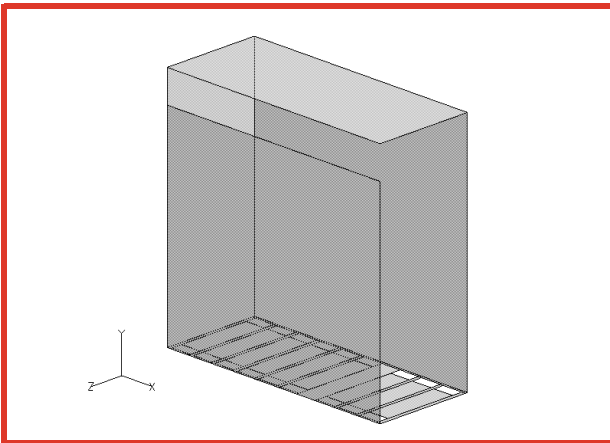
- How to create wall boundaries in your PLOT3D data file
- How to create filled contours and contours on a scalar surface
- How to use Iso-Surfaces to visualize the flame envelope and combustion process
- Using streamlines to investigate air flow patterns
- Visualizing the gas temperature

Step 1: Create Wall Boundary Files

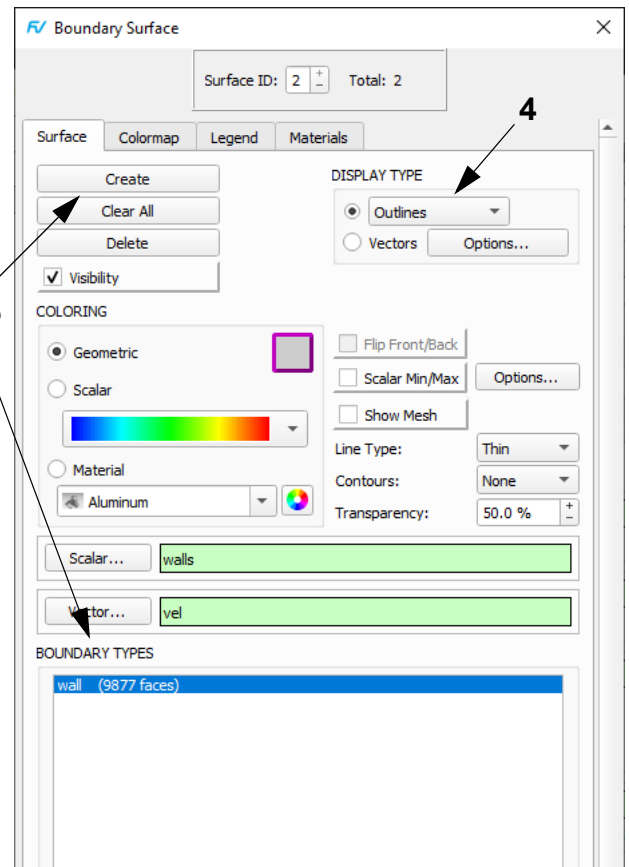
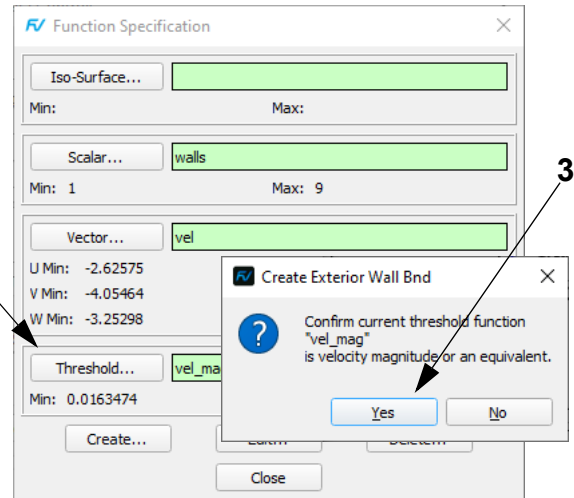
In this dataset, we will look at flow inside a combustor. We will examine several ways to visualize flow features for this type of problem including dataset scaling, subsetting and contours as attributes. Note: Because this tutorial creates a structured boundary file, copy the data to your hard disk before running it.



The pictures on each page correspond to the final image you should see when the steps are completed correctly.

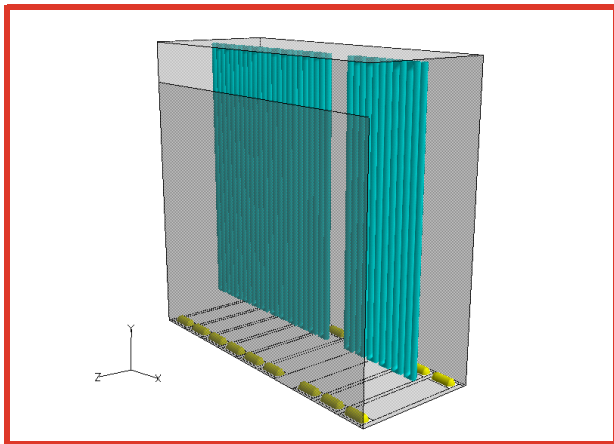


- Bring up the **File**→**Data Input**→**PLOT3D** data reader. Turn **Auto-Detect Format OFF** for this case and use settings *Binary, 3D, no Iblanks or MultiGrid.* Click on the **Read XYZ Data...** button, select the grid file **sw_xyz** and click **Open**. When the **Grid Subset Selection** panel comes up, click **OK**. Click on the **Read Function Data** button, select **sw_fun** and click **Open**. When prompted, choose **Open Function Name File...** select **sw.nam** and click **Open**. Close the **PLOT3D Data Input** panel. [Page 55 Reference Manual]
- Press the **Functions** button on the main toolbar. Press the **Threshold** button and choose **vel_mag** from the **Function Selection** panel. Press **Calculate**, then **Close** the **Function Specification** panel. [Page 100 Reference Manual]
- Create a wall boundary file with the **Tools** → **Create Wall Bnd File** pulldown. This creates a structured boundary file consisting of all grid locations where the **Threshold** function = **0.0**. Press **Yes**. If the message "Boundary type already exists" appears, click **OK**, (re)move the fvbnd file and repeat this step. [Page 516 Reference Manual]
- Bring up the **Boundary Surface** panel. **Create** a surface, select **wall**, then click **OK**. Change the **DISPLAY TYPE** to **Smooth** shading and set **Transparency** to **50%**. [Page 240 Working with FieldView]
- Create** a second **wall** boundary surface. Change **DISPLAY TYPE** from **Smooth** to **Outlines**.
- Go to **Tools** → **Dataset...** On the **Dataset Controls** panel change the **X** scale to **1.2** and the **Z** scale to **1.5**. Press **Apply** when done and **Close**.

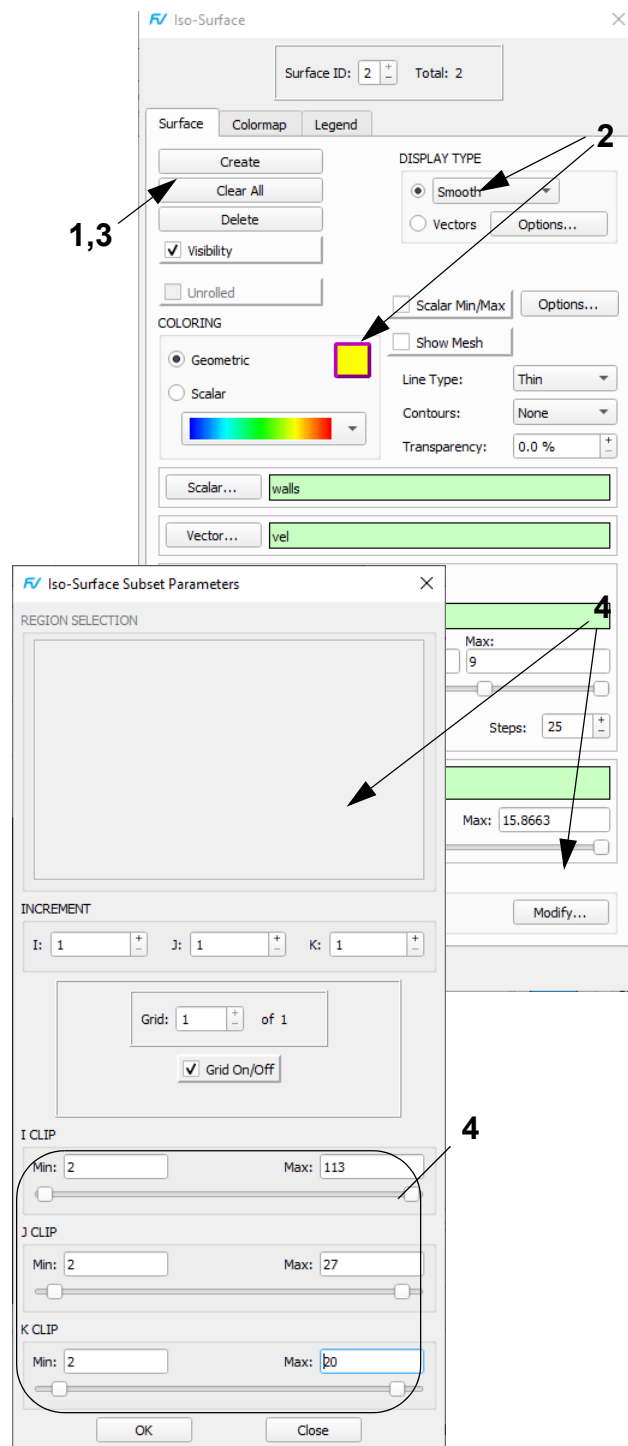


Step 2: Create the Burner Inlet and Tube Coil Walls

In this dataset there is a function called **walls**. It is used to define geometry segments in the model. The **burner inlets** are **walls = 6** and the **tube coils** are **walls = 7** (this also contains the external boundaries). Subsetting will be used to trim these).

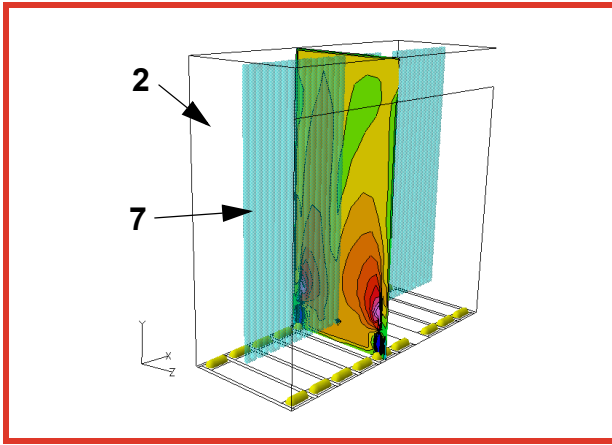


1. Click the **Iso** icon on the side toolbar to bring up the **Iso-Surface** panel. Click the **Create** button. Since this is the first iso-surface we are creating, we will be prompted to select the iso-function we want to use. Click the **Define Iso Function** button, choose **walls** from the **Function Selection** Panel (not shown) as the iso-function and **Calculate**. In the **Current:** field for the Iso Function, enter **6**. [\[Page 170 Working with FieldView\]](#)
2. Change the **DISPLAY TYPE** from **Constant** to **Smooth** shading and the **Geometric** color to yellow. These are the **burner inlets**.
3. **Create** a second iso-surface. In the **Current:** field for the same Iso Function, enter **7**. These are **tube coils** and **external walls**. Change the **Geometric** color to light blue.
4. Since this iso-surface also includes external walls, we will need to subset the surface. Press the **Modify...** button in the lower right corner of the **Iso-Surface** panel. On the **Iso-Surface Subset Parameters** panel, adjust the **Min:** and **Max:** extents in each of the **I**, **J** and **K** directions. Change all three **Min:** values to **2**, and decrease the **Max:** values by one, to **113**, **27** and **20**, respectively. Press **OK** and then **Close**.
5. Reorient the model to approximate the view shown above.

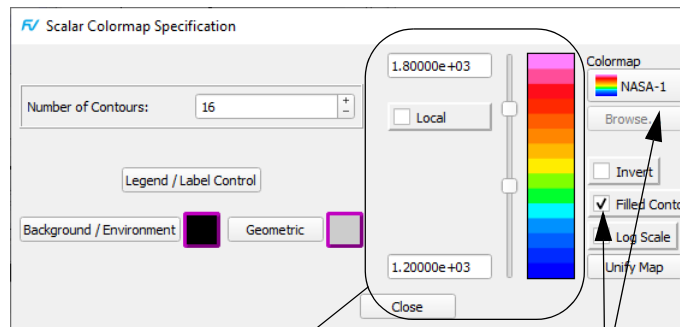
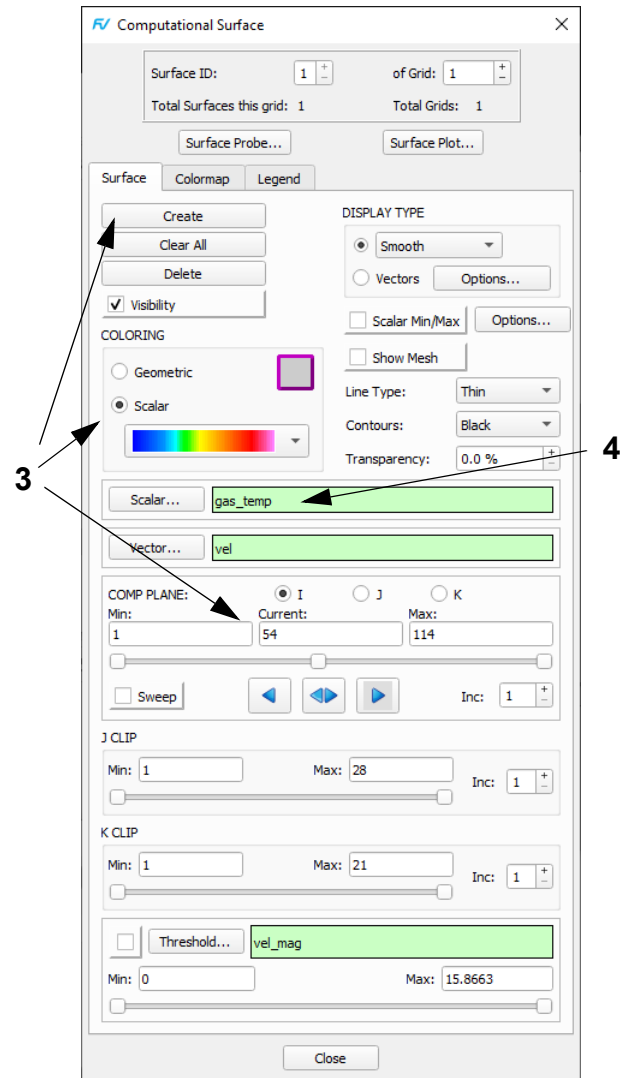


Step 3: Create Filled Contours on a Scalar Surface

In this step we will use the **Filled Contour** feature to help us visualize the gas temperature on a computational surface.

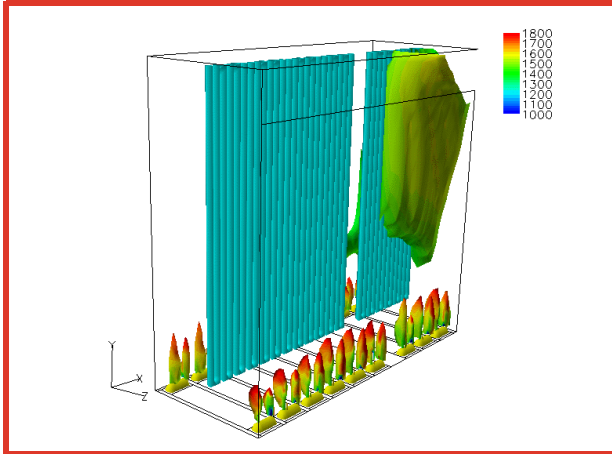


1. Click the Transform Controls toolbar icon to bring up the **Defined Views** panel. Choose an **ISOMETRIC** view of **-X +Y +Z**. Rotate the model as needed to match the view above.
2. Double-click or "Quick-Pick" on the outside wall boundary to bring up the **Boundary Surface** panel for Surface #1. Turn the **Visibility** off (panel not shown). [\[Page 75 User's Guide\]](#)
3. Click the **Comp** icon on the side toolbar to bring up the **Computational Surface** panel. **Create** a computational surface at **I = 54**. Make the surface **Scalar** colored and change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
4. Press the **Scalar...** button for the **Scalar Function** to bring up the **Function Selection** panel. Select **gas_temp** and click **Calculate**. Change the **Contours** setting to **Black**.
5. Click the main toolbar icon to bring up the **Scalar Colormap Specification** panel OR click the **Colormap** tab on the **Computational Surface** panel. Change the minimum and maximum to **1200** and **1800**, respectively. Change the **Colormap** from **Spectrum** to **NASA-1** and check the **Filled Contour** box.
6. The contours will appear more visible with more direct lighting. To reorient the light, change the **Object:** on the **Viewer Toolbar** from **World** to **Light** and rotate the light (actions not shown).
7. "Quick-pick" on the tubes. On the **Iso-Surface** panel, change the **Transparency** to **50%**.

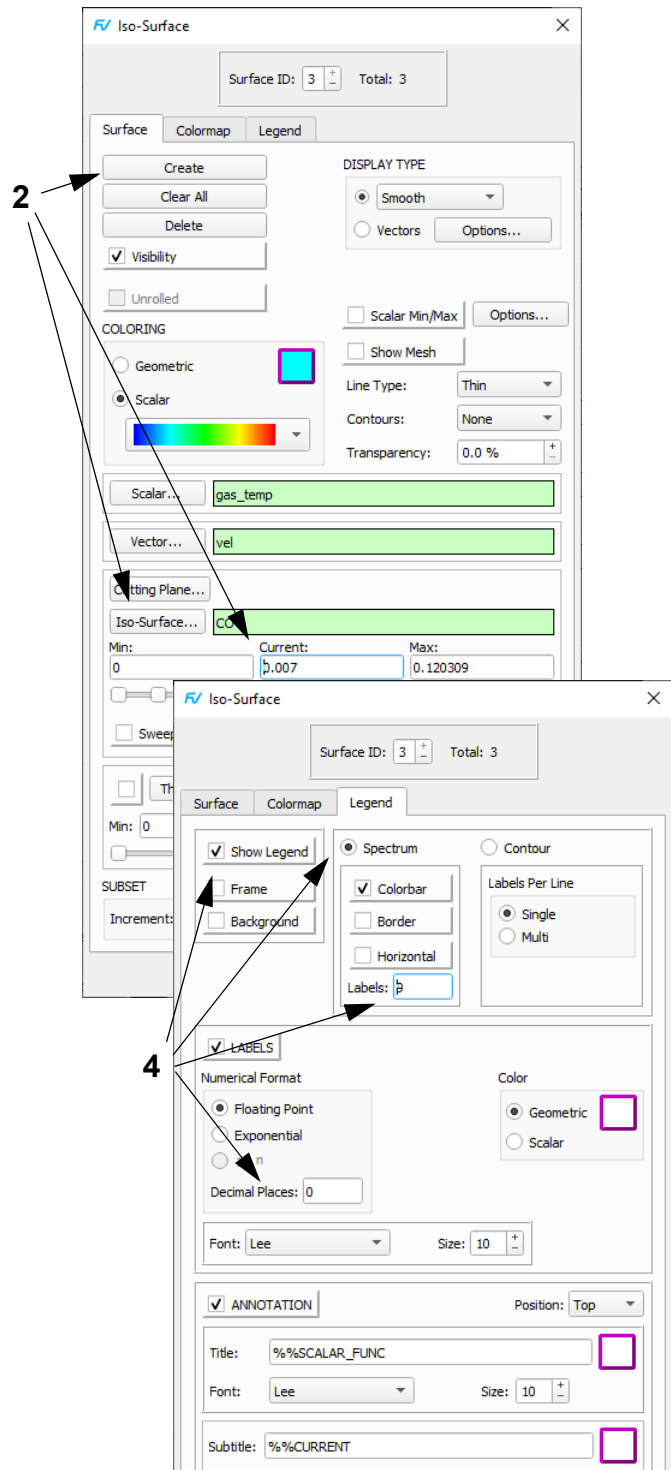


Step 4: Visualize the Flame Envelope

In this step we will use an iso-surface of CO (carbon monoxide) to visualize the flame envelope. The flame is rich in soot, which is being heated to luminosity by the combustion. As soot is formed in fuel-rich regions, concentrations of CO can be used as an indicator.

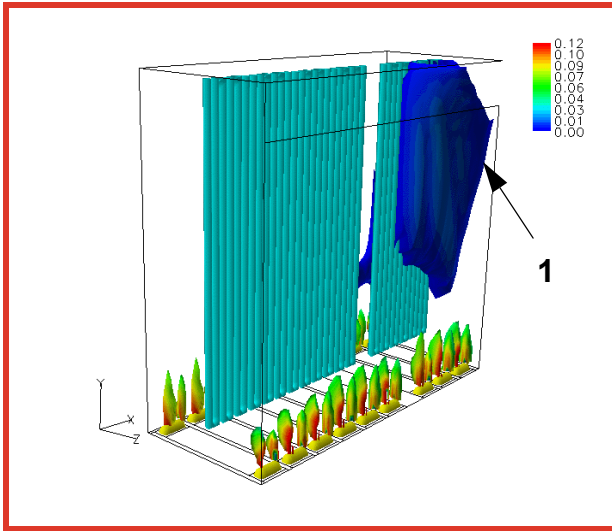


1. "Quick-pick" the computational surface created in the last step and turn the **Visibility** off. "Quick-pick" on the blue tubes and change **Transparency** to **0.0%**.
2. **Create** another iso-surface. Click on the **Iso-Surface...** button for the **Iso Function**. In the **Function Selection** panel, double click on **CO** (double clicking will Calculate the function and close the panel). Change the **Current:** field for the Iso Function to **0.007**. Change the **COLORING** to **Scalar**.
3. On the **Colormap** tab (not shown), adjust the range for **SCALAR COLORING** to **1000** to **1800**. Change the **Number of Contours** to **32**. Change the **Colormap** from **NASA-1** to **Spectrum**. Turn **Filled Contour** off.
4. On the **Legend** tab, turn **Show Legend** on. Create a **Spectrum** legend and adjust the following parameters: **Labels:** to **9** and **Decimal Places:** to **0**.
5. With the mouse pointer in the graphics window, use **SHIFT-Left Mouse** to move and **SHIFT-Right Mouse** to resize the legend as desired.

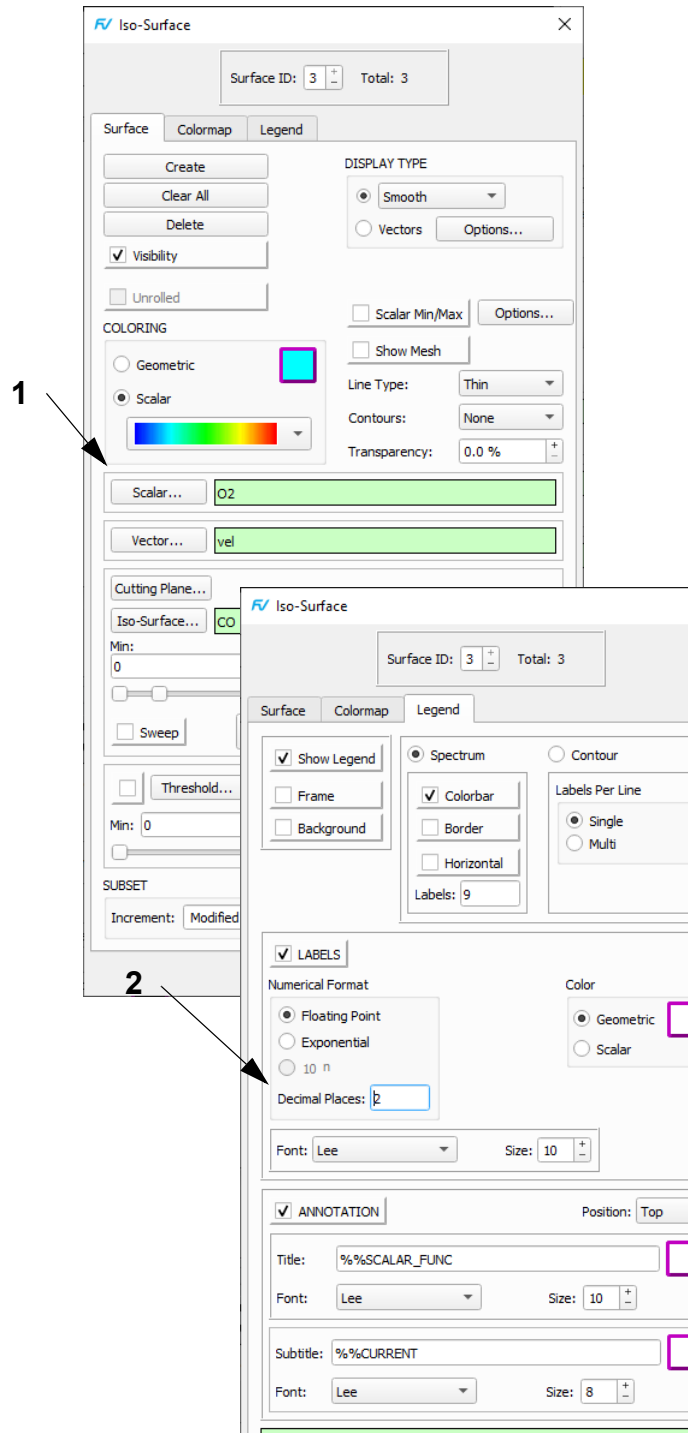


Step 5: Visualize the Combustion Process

In this step we will color the CO (flame envelope) of O₂ (oxygen) to visualize the combustion process. After changing the iso-surface, the large blue region is where the combustion is incomplete; the concentration of O₂ here is very low. Keep in mind that the large region on the right and the “flames” from the burner inlets are all part of the same iso-surface of CO = 0.007.

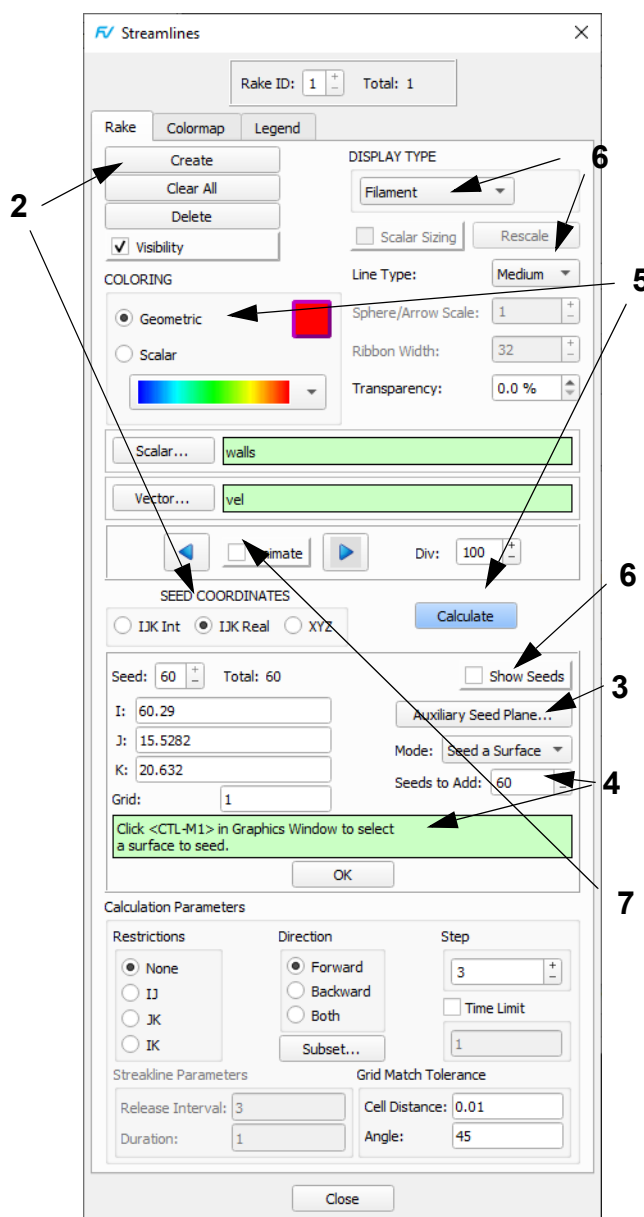
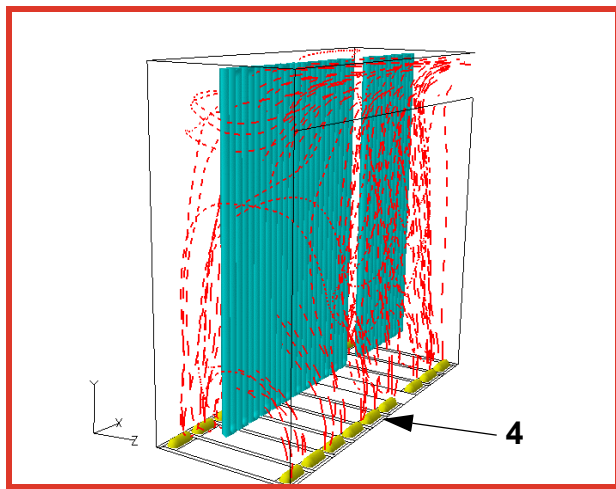


1. Go to the **Surface** tab of the CO Iso-Surface for the flame envelope. Click on the **Scalar...** button for the **Scalar Function**. On the **Function Selection** panel, double click on **O₂** (double clicking will Calculate the function and close the panel).
2. On the **Legend** tab, adjust the number of **Decimal Places** to 2.



Step 6: Use Streamlines to Visualize the Flow Pattern

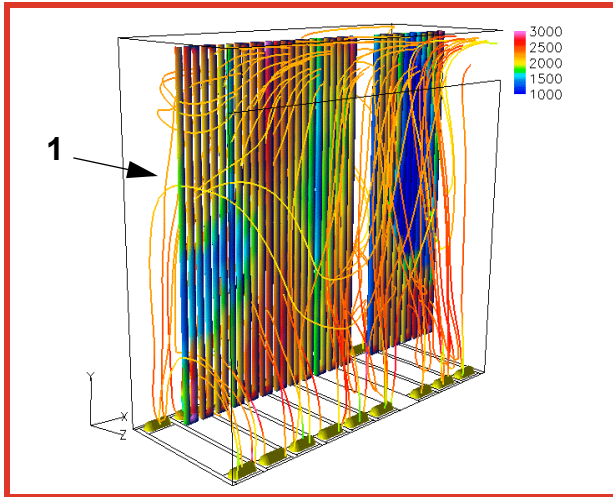
In this step we will use streamlines to show flow patterns in and around the tube coils. Understanding the air-flow patterns within the heater is important because they ultimately dictate the heat flux distribution and the uniformity of the heat transfer to the process tubes. Features in the flow patterns, such as recirculation zones, cause a non-uniform heat transfer to the tubes and walls and may result in uneven heating. An important design goal is to maintain constant heat transfer and exit temperature for the process fluids; one way to accomplish this is to vary the burner firing to compensate for flow patterns and heater geometries. Notice that several of the streamlines pass between the process tubes; expect heat flux to be high in these regions.



1. "Quick-pick" the **CO** iso-surface (Surface ID 3) and turn the **Visibility** off.
2. Bring up the **Streamlines** panel. Press **Create** and change the **SEED COORDINATES** from **IJK Int** to **IJK Real**. [Page 184 Working with FieldView]
3. Change the **Mode** from **Add** to **Seed a Surface**.
4. Pick on one of the yellow burners with the mouse using **CTRL-M1**, following the hint on the **Streamlines** panel. Adjust the **Seeds to Add**: value from **10** to **60**. Press **OK** to add the seeds.
5. Change the **Geometric** color to red and press **Calculate**.
6. Turn off **Show Seeds**. Change the **DISPLAY TYPE** from **Complete** to **Filament**. Change the **Line Type** to **Medium**.
7. Turn **Animate** on. While the streamlines are animating, rotate the dataset to gain a better understanding of the flow patterns.
8. Turn **Animate** off before proceeding to the next step.

Step 7: Visualize the Gas Temperature

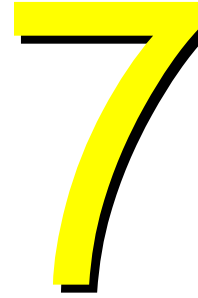
In this step we will use the “Function Calculator” to create a function of **gas_temp** that will be proportional to the heat flux. Note how the streamlines from the burners, now colored with temperature, pass between the tube coils at the higher heat flux points. This is consistent - heat flux ties in the convective flow and the higher temperature fluid coming into contact with the coils.



1. For the streamline rake, change the **DISPLAY TYPE** to **Complete** and the **COLORING** to **Scalar**.
2. Click the **Scalar...** button to select the **Scalar Function**. Double click on **gas_temp** on the **Function Selection** panel.
3. On the **Colormap** tab, change the current **Colormap** from **Spectrum** to **NASA-1**. Change the **SCALAR COLORING** range to **1000** to **1800**.
4. Press the **Function Specification** icon on the main toolbar. On the **Function Specification** panel, click **Create....** In the **Function Formula Specification** panel, create the **Formula: mag(grad("gas_temp"))**. Click **OK** and **Use Formula** for the function name.
5. “Quick-pick” the blue tube coils. Change the **COLORING** to **Scalar**. Change the **Scalar Function** to **mag(grad("gas_temp"))** from the **Function Selection** panel. On the **Colormap** tab, change the range of the colormap to **1000** to **3000**.
6. On the **Legend** tab, create a legend by turning **Show Legend** on. Specify **5 Labels** and **0 Decimal Places**.

4

Chapter 7



Basic Turbomachinery Tutorial

What You Will Learn

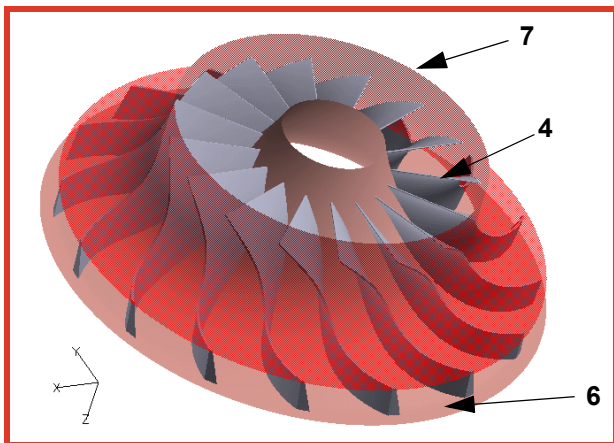
- How to rotate a dataset with periodic boundary conditions
- How to use an iso-surface to show flow inefficiencies
- How to use an iso-surface for streamline seeding
- How to display streamlines downstream and upstream of the streamline seeds
- How to examine flow leakage
- How to display streamlines as particle paths
- How to use the Function Panel to display radial pressure contours
- Creating and "Quick-Picking" the annotation
- How to use "Sweep Integration" and display the results in a 2D Plot

Step 1: Working with Turbomachinery Data

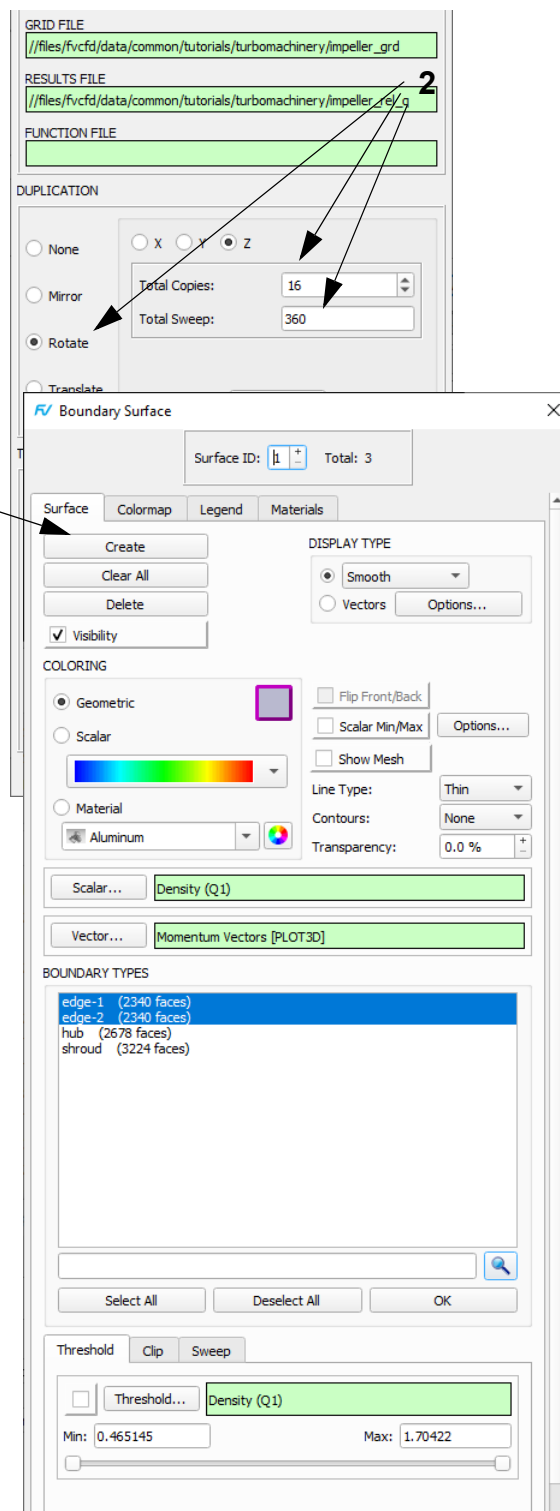
In this tutorial we look at a geometrically periodic dataset with **FieldView**-generated rotational copies. We will use periodic streamlines to show flow blockage, leakage and to compute mass flow.



The pictures on each page correspond to the final image you should see when the steps are completed correctly.

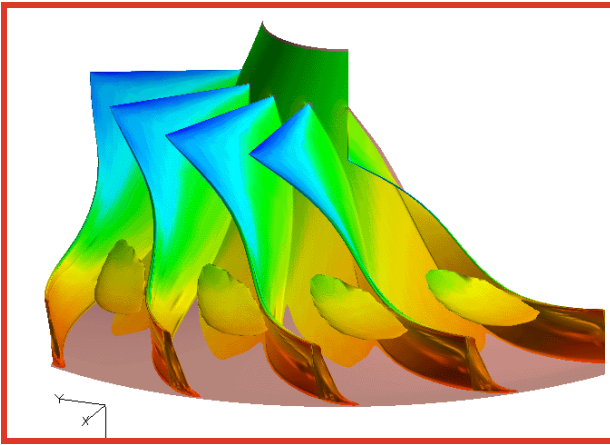


1. Choose **File** → **Data Input** → **PLOT3D**. Read XYZ Data **impeller_grd** and Read Q Data **impeller_rel_q** using the default setting **Auto Detect format** (actions not shown). [Page 55 Reference Manual]
2. Bring up the **Dataset Controls** panel from **Tools** → **Dataset...** and turn on the **Rotate** button in the **DUPLICATION** section. In the **Dataset Rotate Parameters** panel, set **Total Copies:** to 16, leave **Total Sweep:** 360 and set the axis of rotation to **Z**. Press **OK** and **Close** the two panels.
3. Bring up the **Boundary Surface** panel and **Create** a surface by selecting **BOUNDARY TYPES edge1** and **edge2**. Press **OK**. Change the **DISPLAY TYPE** to **Smooth** shading. Change the **Geometric** color to green by clicking on the color chip and selecting green from the **Geometry Color** panel.
4. **Create** a second boundary surface, selecting the **BOUNDARY TYPE hub** with **Geometric** color yellow.
5. **Create** a third boundary surface with the **shroud** with the **Geometric** color red. Set **Transparency** to 50%.
6. From the **View** pulldown menu, turn off **Outline**.
7. From the **Tools** pulldown, bring up the **Color Mixer** panel. Click on the green color chip and change the **Red, Green** and **Blue** values to 185, 185, and 207. Press **Apply**. Click on the yellow chip and change the **Red, Green** and **Blue** values to 225, 160, and 152. Press **Apply** again and **Close** the panel.
8. Transform the dataset to the view illustrated above.

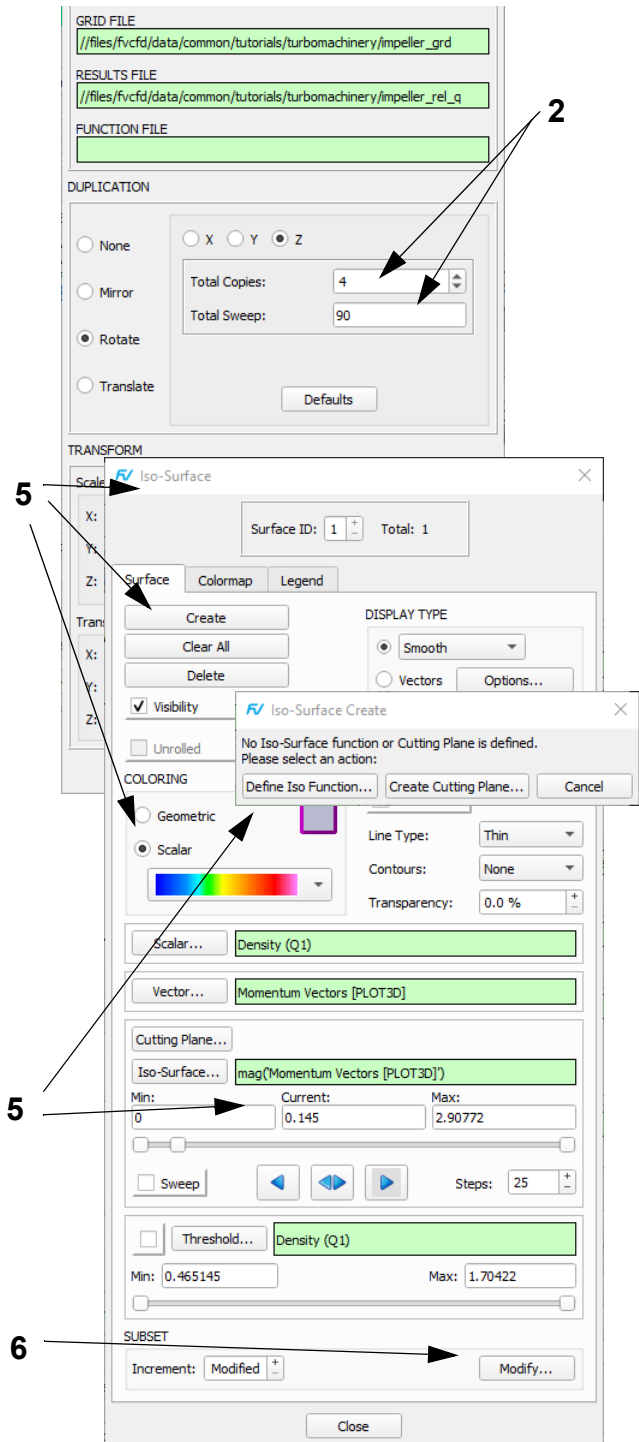


Step 2: Use an Iso-Surface to Show Flow Blockage

An iso-surface will be generated to show flow blockage of the impeller. This iso-surface will be used as a streamline seed surface in the next step.

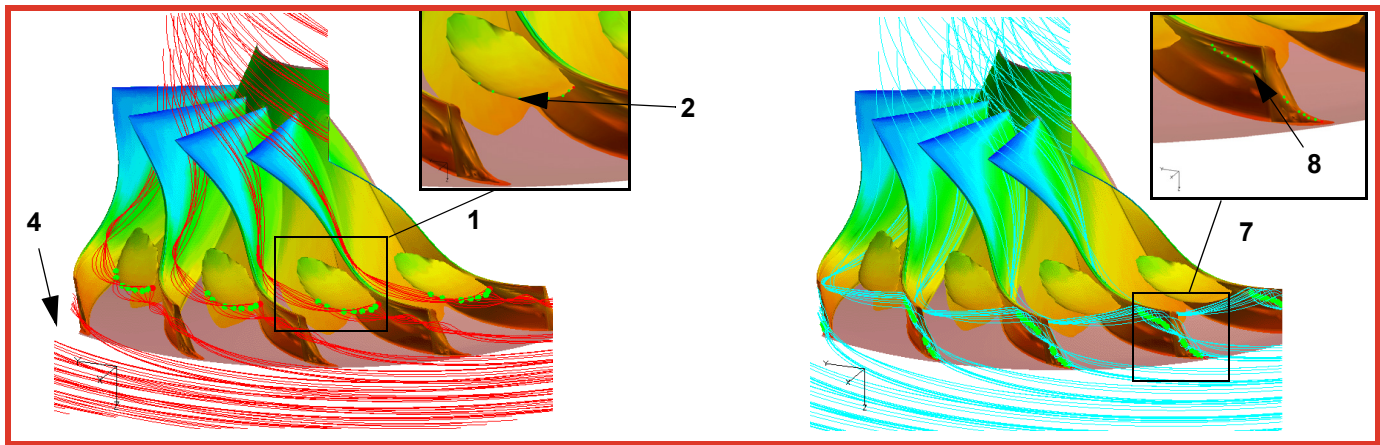


1. Quick-pick (double-click) the transparent **shroud** boundary surface. Turn the **Visibility** off (not shown). [\[Page 113 Working with FieldView\]](#)
2. Bring up the **Dataset Controls** panel, press **Parameters...**, adjust **Total Copies: 4** and **Total Sweep: 90** and click **OK**. Close the **Dataset Rotate Parameters** and the **Dataset Controls** panels.
3. Using the mouse, zoom in and reposition the view to that shown above.
4. Press the **Functions** button on the main toolbar and. Press **Create** on the Function Specification panel. Using the **Function Formula Specification** panel, create the function: **mag("Momentum Vectors [PLOT3D]")**. Click **OK** and **Use Formula** for the function name. [\[Page 100 Reference Manual\]](#)
5. Bring up the **Iso-Surface** panel and press **Create**. Press **Define Iso Function...** and select the function created in step #4, which appears at the bottom of the list. Press **Calculate**. Change the **Current** value to **0.145**, **COLORING** to **Scalar** and **DISPLAY TYPE** to **Smooth** shading. [\[Page 170 Working with Field-View\]](#)
6. Press the **Modify...** button to bring up the **Iso-Surface Subset Parameters** panel. Change the **Min:** value of **K CLIP** to **2**. Press **OK** for the change to take effect, then **Close** the **Iso-Surface Subset Parameters** panel.
7. On the **Colormap** tab [\[Page 52 Working with Field-View\]](#), change the **Colormap** from **Spectrum** to **NASA-1** (not shown).

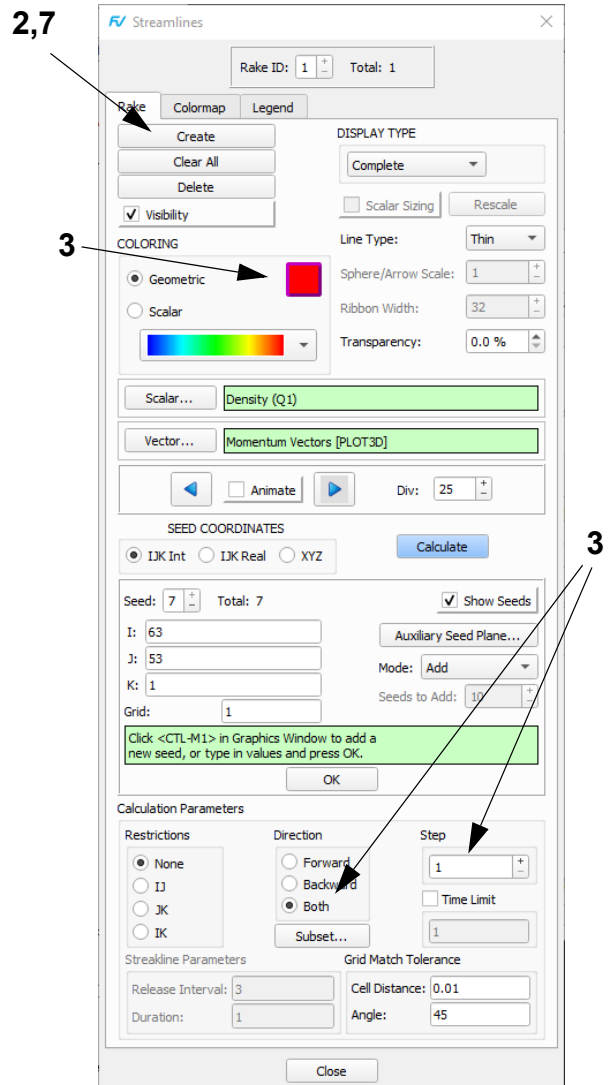


Step 3: Streamline Seeding Using an Iso-Surface

The blockage iso-surface will be used for streamline seeding.



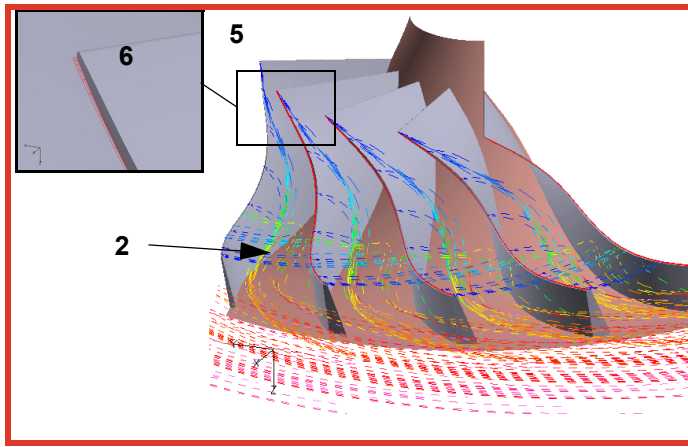
1. Zoom into one of the iso-surfaces as shown in the inset image at the upper left by clicking the **Zoom** icon on the side toolbar, using the left mouse (M1) to define a zoom area.
2. Bring up the **Streamlines** panel and press **Create**. Change the **SEED COORDINATES** from **IJK Int** to **IJK Real**. Add 8 seeds along the iso-surface edge (see inset) using CTRL-M1 to define seed locations. [\[Page 184 Working with FieldView\]](#)
3. Change the **Calculation Parameters** as follows: Change **Direction** of the streamline integration from **Forward** to **Both** and the integration **Step** number to **9**. Change the **Geometric** color to red.
4. Press **Calculate**. Press the **Undo Zoom** icon on the side toolbar to zoom back out. Notice the complex flow patterns made visible by the streamlines. We see the vortex flow near the outer section of the blades. We have used the Momentum Vectors [PLOT3D] as the streamline vector function. Using the Velocity Vectors [PLOT3D] results in minimal change.
5. To make the creation of the next streamline rake easier, turn the **Visibility** of the first rake off.
6. Zoom into one of the iso-surface portions as shown in the inset image at the upper right by clicking the **Zoom** icon on the side toolbar, using the left mouse (M1) to define a zoom area.
7. Press **Create** on the **Rake** tab again. Put 8 seeds along the upper ridge of the iso-surface near the outer blade surface. Press **Calculate** and **Undo Zoom**. Change the **Geometric** color to light blue.



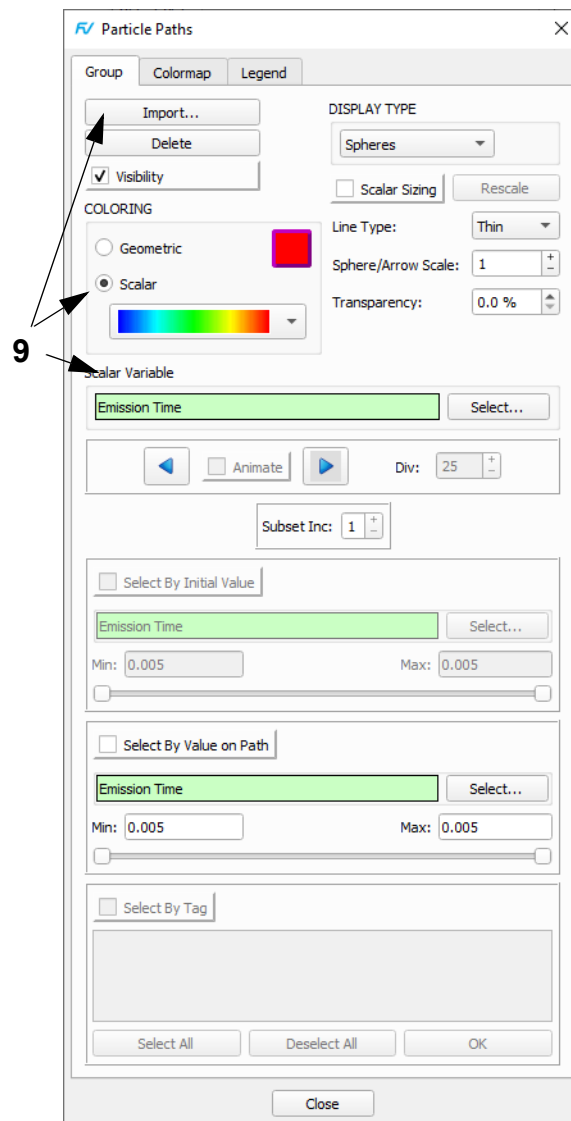
Step 4: Visualize Flow Leakage

In this step we will use subsetting of a computational surface and streamline export (and subsequent import as particle paths) to show the flow leakage between the blades and the shroud.

along these paths. [\[Page 215 Working with FieldView\]](#)

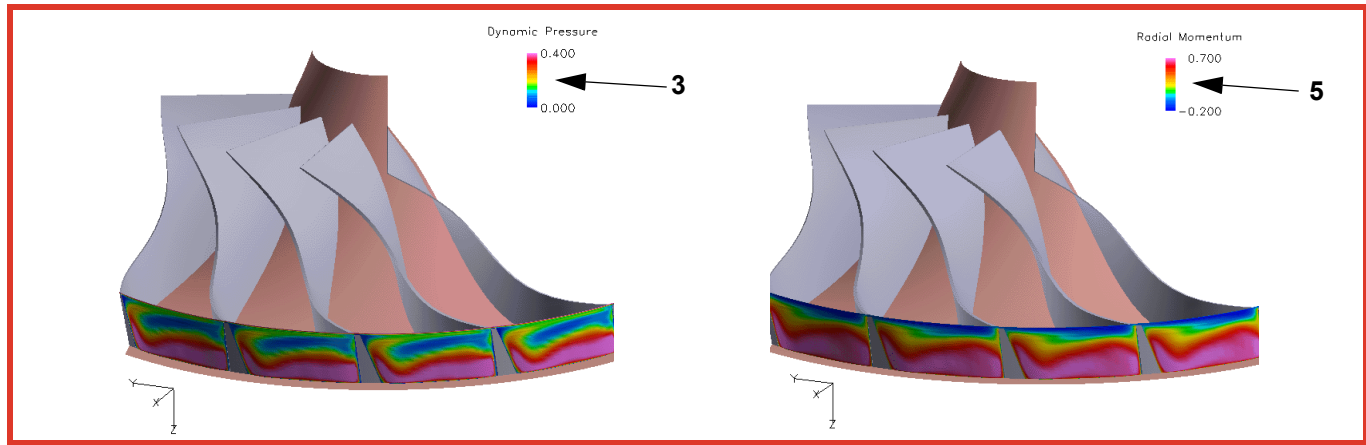


1. Turn the **Visibility** and **Display Seeds** of both streamline rakes off.
2. Quick-pick the iso-surface and turn **Visibility** off.
3. Bring up the **Computational Surface** panel (panel not shown). **Create** a **J=53** surface. Change the **I Min:** and **Max:** to **40** and **70**, and the **K Min:** and **Max:** to **41** and **43**, respectively. Change the **Geometric COLORING** to magenta and the **DISPLAY TYPE** to **Constant** shading. [\[Page 161 Working with FieldView\]](#)
4. Zoom into the computational surface as shown in the inset image above. We will use the Computational Surface from step #3 for streamline seeding.
5. Bring up the **Streamlines** panel (panel not shown). Create a third rake. Change **SEED COORDINATES** to **IJK Int.** Change **Mode** to **Seed a Surface**. Use the default value of **Inc: 3** and turn on **Show Seeds**. Carefully pick the magenta surface with **CTRL-M1**. Press **OK**.
6. Change the **Calculation Parameters** as follows: Change **Direction** to **Forward**. Turn on **Time Limit** and enter **0.2** to “clamp” the duration of the streamlines to a maximum of **0.2** time units. Press **Calculate**.
7. Bring up **Tools** → **Export** and save the streamlines as **FieldView** particle paths by using **.fvp** as the file extension.
8. Turn the **Visibility** and **Show Seeds** of this third streamline rake off.
9. Bring up the **Particle Paths** panel and **Import** the exported streamlines. Change **COLORING** to **Scalar** and set the **Scalar Variable** to Duration to show time



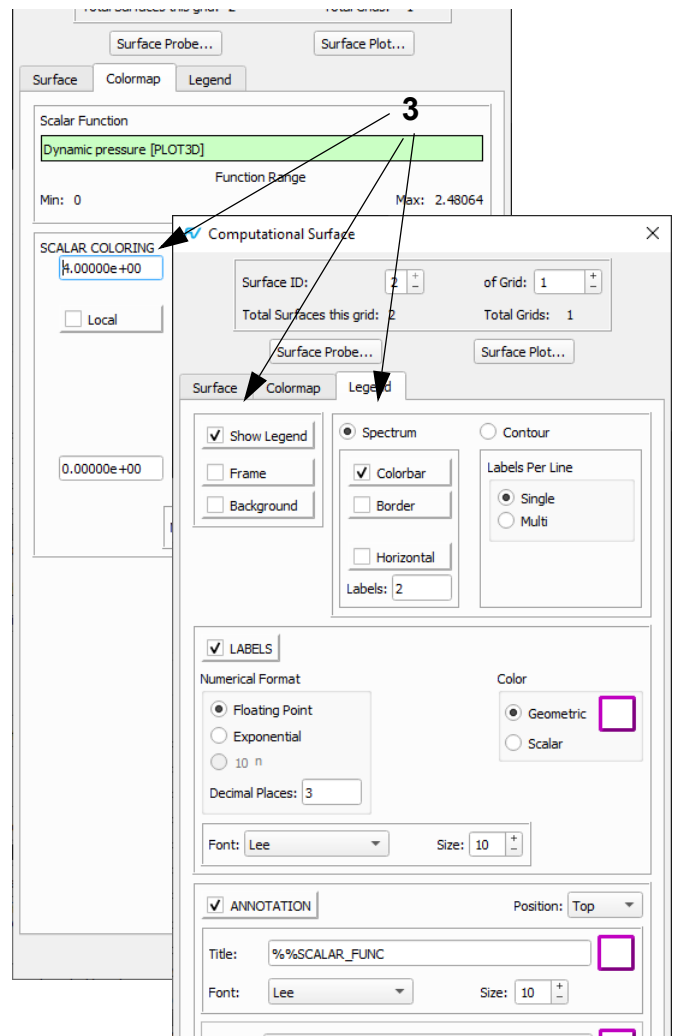
Step 5: Create Radial Pressure Contours

In this step we will create a computational surface at the end of the blade passage to show contours of Dynamic Pressure and Radial Momentum.



1. Turn off **Animate** and the **Visibility** of the particle paths and magenta computational surface (action not shown). Orient the model similar to the above view.
2. **Create** a second computational surface at **I=72**. Since it inherits the previous surface properties, you'll need to set the **Max:** value greater than **72** (it will say **70**). Change **Min:** and **Max:** of **K** back to **1** and **47**. Change **COLORING** to **Scalar** and **DISPLAY TYPE** to **Smooth** shading. Change the **Scalar Function** by pressing the **Scalar...** button to bring up the **Function Selection** panel. Double click on **Dynamic Pressure [PLOT3D]** to select it and close the panel.
3. On the **Colormap** tab, change the maximum to **0.4**. On the **Legend** tab, turn on **Show Legend**. Check to make sure the **Spectrum** button is selected. Modify your legend Title & Subtitle as desired. Use the directions at the bottom of the panel to move and scale the legend.
4. For the second image (at right), create a new function, radial momentum (see expression below), using the **Function Formula Specification** panel. This formula will use some of the built-in functions available with **FieldView**.

$$U_r = "X"/"Rcyl: (X^2+Y^2)^{.5}" * "x-momentum (Q2)" + "Y"/"Rcyl: (X^2+Y^2)^{.5}" * "y-momentum (Q3)"$$
5. Quick-pick (double-click) on the computational surface to make sure it is current. Change the **Scalar Function** to **Ur**. Adjust the **Colormap** range to **-0.2** to **0.7**. Note that the legend is automatically updated to reflect these changes.



Chapter 8

8

Vortex Shedding Tutorial

What You Will Learn

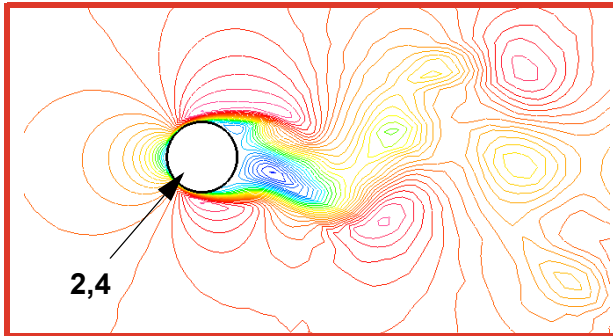
- [How to read in a FieldView Unstructured Transient Data File](#)
- [How to animate a scalar surface of pressure by conducting a transient sweep](#)
- [How to create a streamline seeding surface](#)
- [How to set the streamline Time Limit](#)
- [How to make a movie of a transient streakline](#)
- [How to display the time step and solution time of a transient sweep](#)
- [Importing streaklines and improving the animation](#)
- [How to use scripts and create DataGuide files](#)

Step 1: Read a Transient Dataset

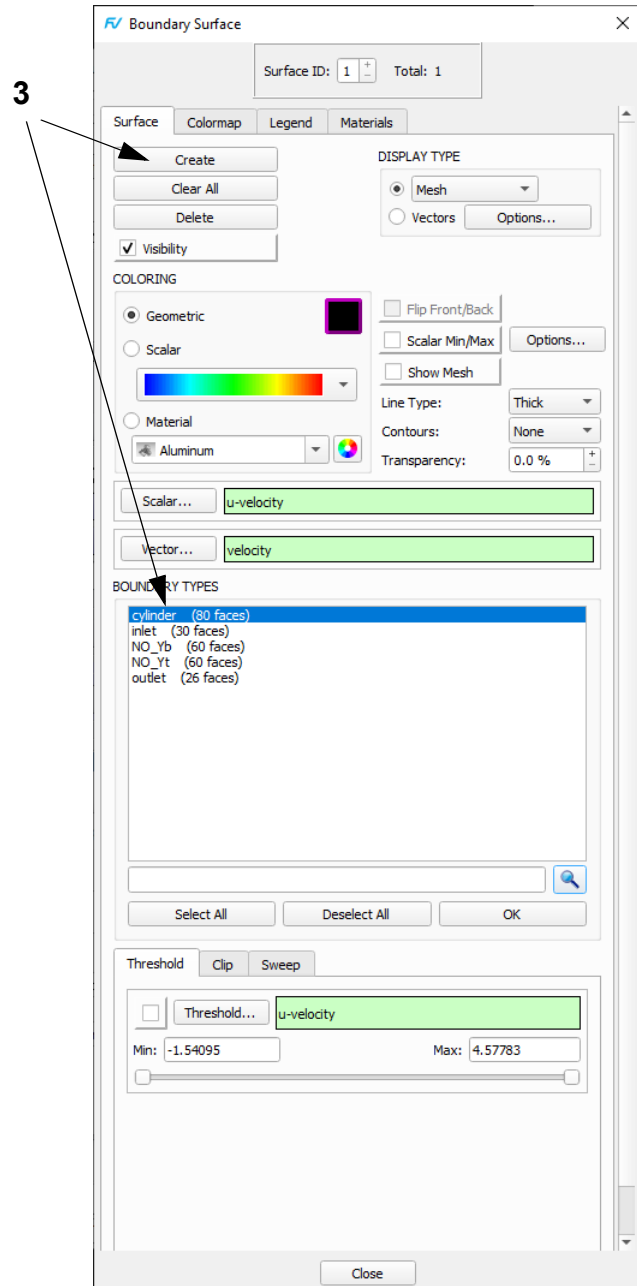
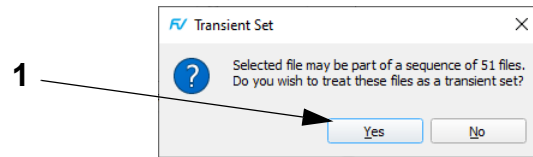
This tutorial will show the user how to work with transient data. We will also show how to create streaklines to visualize transient flow patterns. An outline is presented for setting up rakes which can be used for subsequent work with other datasets.



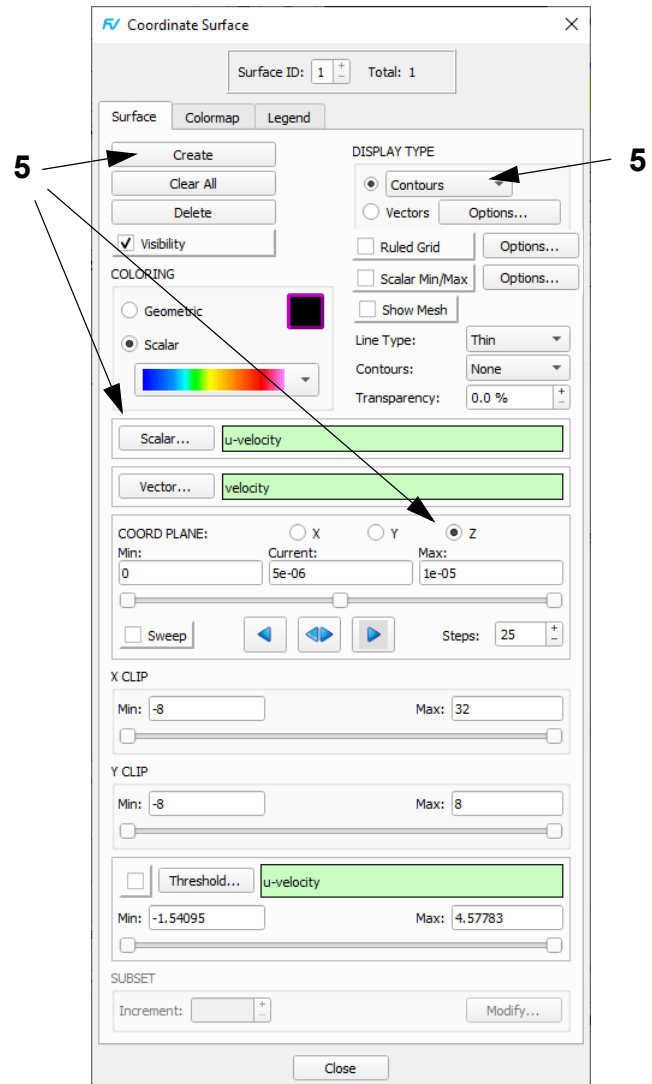
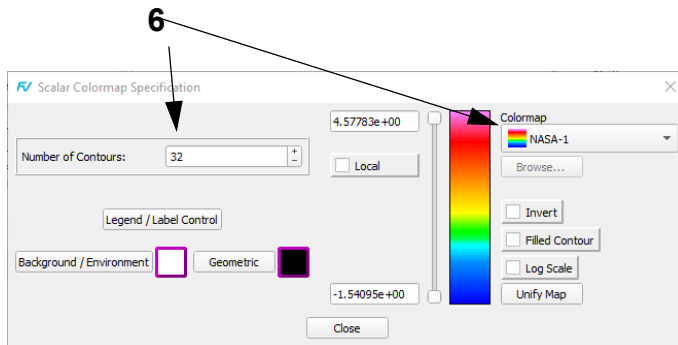
The pictures on each page correspond to the final image you should see when the steps are completed correctly.



1. Bring up **File** → **Data Input** → **AcuSolve [FV-UNS Export]**, click **Read Grid or Combined Data...**, select one of the files in the **FieldView** Unstructured transient dataset called **shed_000.uns,...,shed_050.uns** and **Open**. Click **Yes** to treat all 51 files as a transient set. After read in, you will be presented with an iso-metric view of the data outline. **Close** the **AcuSolve [FV-UNS Export]** data input panel. [\[Page 37 Reference Manual\]](#)
2. Use the Transform Controls toolbar icon OR **View** → **Defined Views** to bring up the **Defined Views** panel (panel not shown). Click the **+Z VIEWING DIRECTION**. Click the Transform Controls toolbar **Center** icon. Move the model to the left and zoom in the graphics window similar to that above. **Close** the **Defined Views** panel.
3. To visualize the cylinder, bring up the **Boundary Surface** panel. Click **Create**, select **cylinder** and click **OK**. Change **Line Type** to **Thick**. Change the **Geometric** color to black.
4. Zoom into the cylinder (with **Object: World** on the **Viewer Toolbar**) with M3 until it is approximately the size shown in the illustration above.
5. To visualize the vortex shedding, **Create** a coordinate surface. Change the **COORD PLANE** to **Z**. Change the **COLORING** to **Scalar** and the **DISPLAY TYPE** from **Mesh** to **Contours**. [\[Page 232 Working with FieldView\]](#)

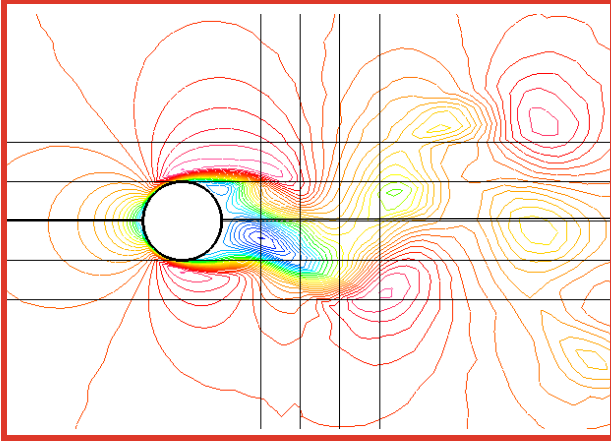


6. On the **Scalar Colormap Specification** panel OR the **Colormap** tab, change the **Number of Contours** to **32** and the **Colormap** from **Spectrum** to **NASA-1**.
7. On the **View** menu (OR the **Viewer Options** panel via the **View** menu OR **Viewer Toolbar** button), turn off the **Axis Markers** and **Perspective**.

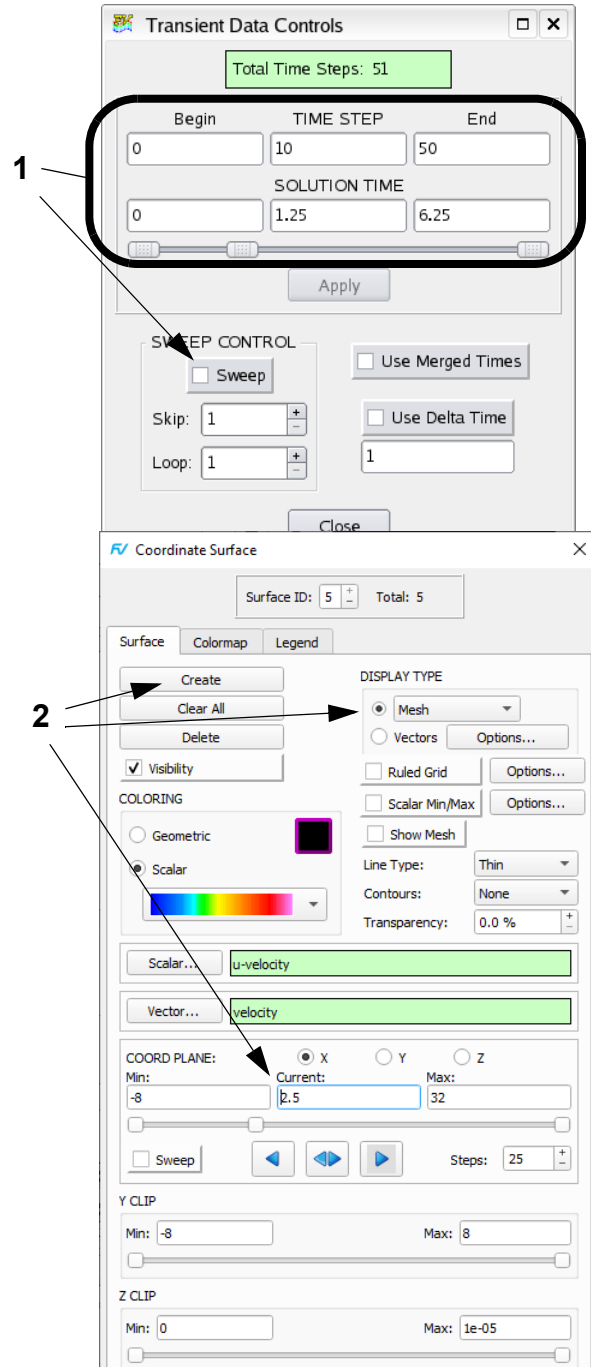


Step 2: Perform a Transient Sweep

In this step, we will watch the movement of the vortex “tail” as we perform a transient sweep and create a grid of coordinate surfaces in order to better see the extent of the movement.

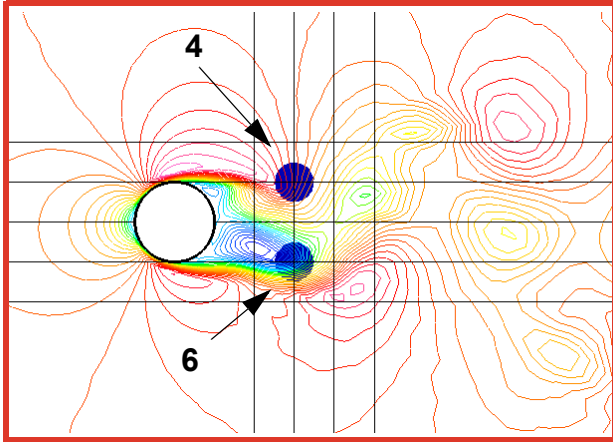


1. Go to **Tools** → **Transient Data...** On the **Transient Data Controls** panel, move the slider to **TIME STEP 0** and click **Apply**. Turn on **Sweep** and watch the blue contours of the vortex “tail” sweep up and down. Notice that the vertical extent of the “tail” is approximately that of the unit cylinder, and note that the center of the cylinder is the point **[0,0,0]**. This information will be used to create a “grid” in the following steps (panel shown has been captured during the sweep, not at the beginning when **TIME STEP** is 0). Click **Sweep** again to stop it. Reset the current **TIME STEP** to 0, click **Apply** and **Close** the **Transient Data Controls** panel.
2. We will create **X** and **Y** planes perpendicular to the screen, which show up as “grid lines”. On the **Coordinate Surface** panel, click **Create**, change **COLORING** to **Geometric**, **DISPLAY TYPE** to **Mesh**, **COORD PLANE** to **X** and the **Current** value to 1.0. Create additional vertical **X** coordinate surfaces at 1.5, 2.0, and 2.5. These will be coordinate surface numbers 3-5 (panel reflects 5th surface).
3. Create horizontal **Y** coordinate surfaces at 0.0, 0.5, 1.0, and at -0.5 and -1.0. These will be coordinate surface numbers 6-10 (surfaces not represented in panel to right).
4. Use the **Transient Data Controls** to **Sweep** through the time steps again. Notice how the blue contoured “tail” sweeps through the points **[1.5, 0.5]** and **[1.5, -0.5]**. These two points will be used to create small coordinate surfaces to seed onto in the next step.



Step 3: Create Streamline Seeding Surfaces

In this step, we will create seeding surfaces at the vertical extents of the “tail” swing. We will create two circles by thresholding coordinate surfaces. First we will create the two threshing functions, circles centered on [1.5, 0.5] and [1.5, -0.5].



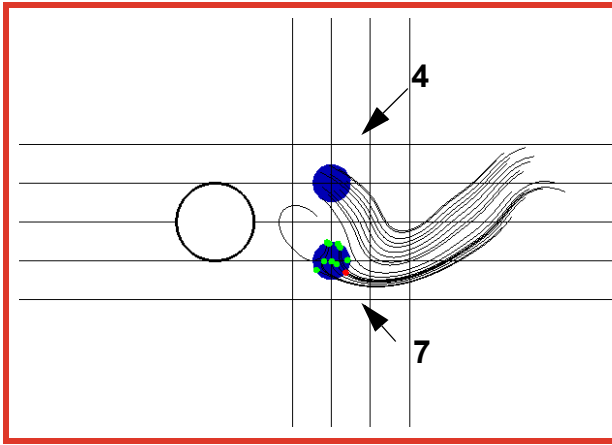
1. Click the **Functions** icon on the main toolbar. On the **Function Specification** panel, click **Create**. On the **Function Formula Specification** panel (“Function Calculator”), create the **Formula**: $((X-1.5)^2 + (Y-0.5)^2)^{0.5}$ by clicking the **Operations/Keys** and selecting the **X** and **Y** Quantities. Click **OK**. [Page 100 Reference Manual]
2. In the **Function Name Input** panel, name the function **Center-1** and click **OK**.
3. Create the **Formula**: $((X-1.5)^2 + (Y+0.5)^2)^{0.5}$, click **OK**, name it **Center-2** and click **OK** (action not shown).
4. Create a **Z** coordinate surface and change the **DISPLAY TYPE** to **Constant** shading. For the **Threshold Function**, click **Threshold...**, choose **Center-1** from the **Function Selection** panel and **Calculate**.
5. Turn on **Threshold Clip** and change **Max:** to **0.25**. Change the **Geometric** color of the surface to blue.
6. Create another **Z** coordinate surface. This surface will inherit most of the properties of the previous surface. Change the **Threshold Function** to **Center-2**. Change **Max:** to **0.25**.

1

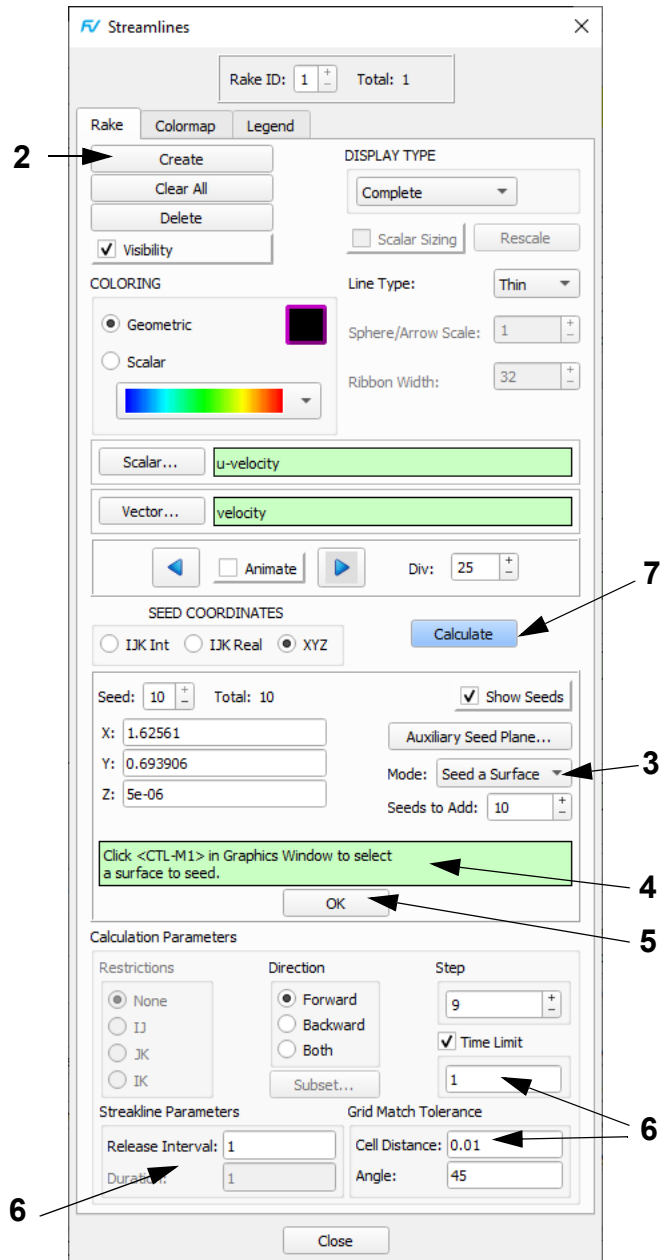
2

Step 4: Create Streamlines

In this step, we will seed the thresholded coordinate surfaces from the previous step. This will be easier by turning off the visibility of the contour coordinate surface.

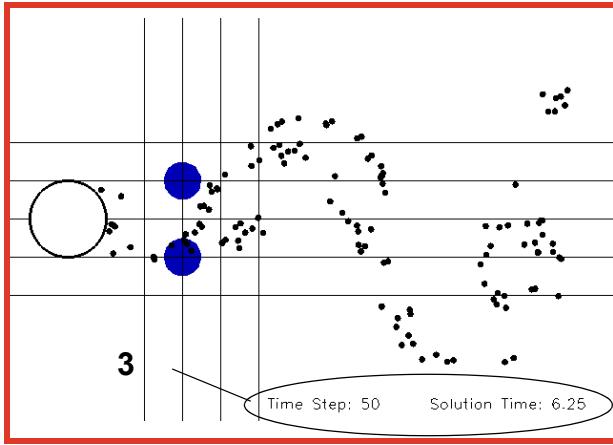


1. Quick-pick the contours to bring up the **Coordinate Surface** panel (**Surface ID: 1**) and turn **Visibility** off (action not shown).
2. Click the side toolbar icon or use the **Visualization Panels** menu to bring up the **Streamlines** panel. **Create** a streamline rake. [Page 184 Working with Field-View]
3. Change the seeding **Mode** from **Add** (default) to **Seed a Surface**.
4. With the mouse pointer the upper of the two blue coordinate surfaces (avoid clicking on the grid), press **CTRL-M1** to choose the surface. An "x" should appear on the selected surface.
5. Press the **OK** button to add the 10 seeds.
6. Make the following changes to the **Calculation Parameters** at the bottom of the **Streamlines** panel: Turn **Time Limit** on using the default value of 1 to limit the duration of all streamlines in this rake to a maximum of 1. Increase the **Step** size from the default of 3 to 9. In the **Streakline Parameters** section, change the **Release Interval** to 1. This will be used in streakline calculations later on.
7. Press **Calculate** to show the time limited streamlines. Note: The appearance of the streamline pattern will depend on the current time step. The above image is the result for time step 0.
8. **Create** a second rake (not shown) and repeat steps 4 through 7 as necessary for the other thresholded coordinate surface.

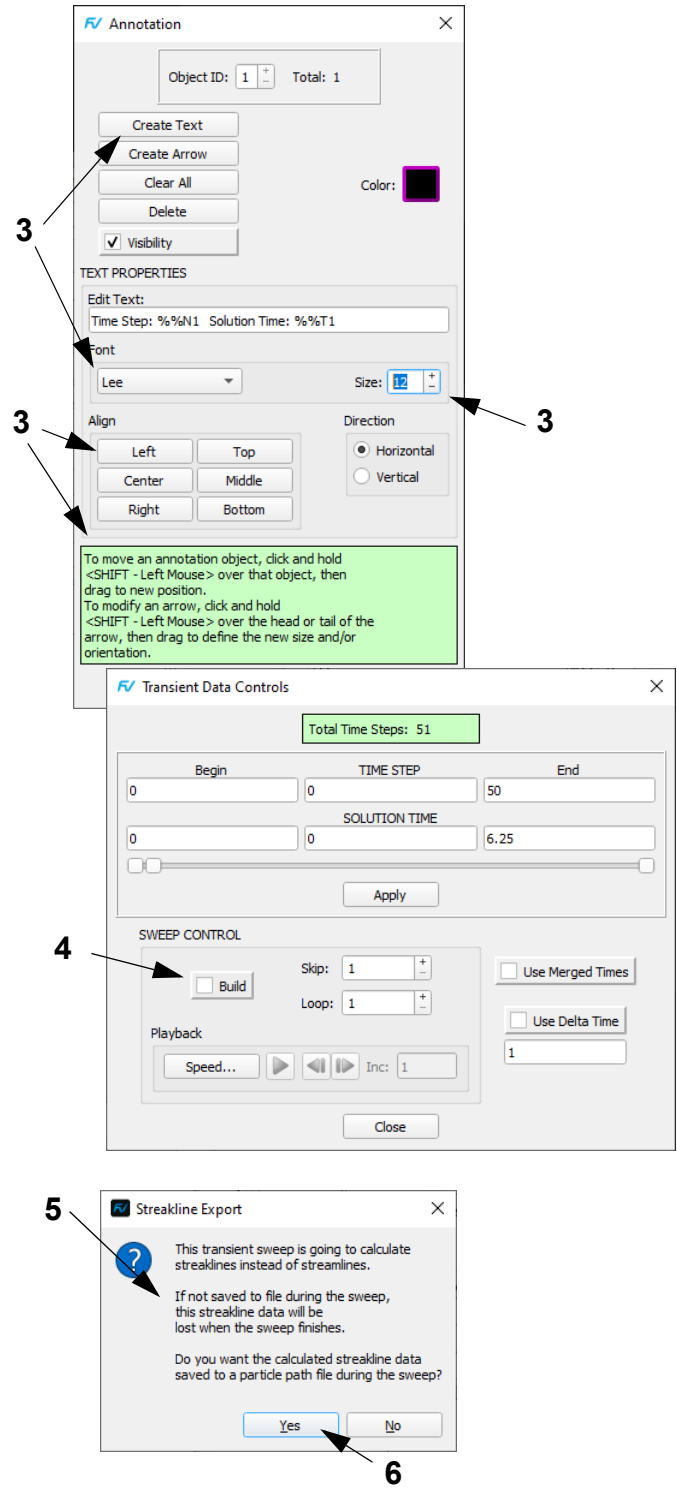


Step 5: Create Streaklines

In this step, we will sweep through the transient dataset. While doing this, we will create a streakline export file and a flipbook (movie) of the particles.

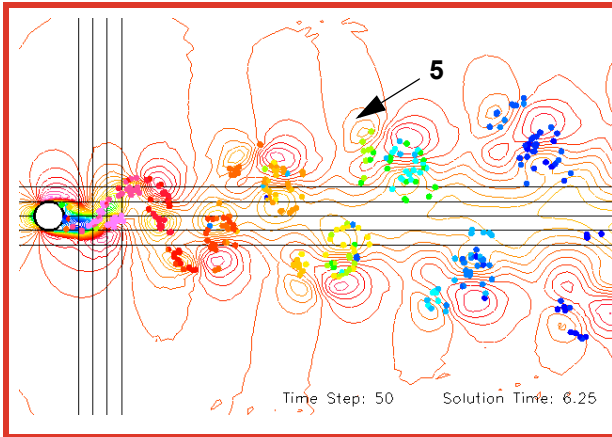


1. Turn off **Show Seeds** for both rakes (not shown).
2. Zoom out and move the cylinder to the left edge of the graphics window to better see the vortices being propagated downstream of the cylinder.
3. Choose **Visualization Panels**→**Annotation** OR click the **Anno** icon on the side toolbar. To create titles showing the time step and solution time being displayed, click **Create Text**. In the **Annotation Create** panel, enter in the string: **Time Step: %%N1 Solution Time: %%T1** and click **OK**. The special notation **%%N1** means show the time step of dataset #1 and **%%T1** means show the solution time of dataset #1. [\[Page 224 Working with FieldView\]](#) Keep the default font, **Lee**, and increase the **Size** as desired. Move the title with the SHIFT+left mouse (M1) following the hints on the **Annotation** panel.
4. Navigate to **Tools** → **Flipbook Build Mode** to prepare to create a flipbook. Click **OK** to dismiss the **Flipbook Display Size Warning** panel (not shown). Bring up **Tools** → **Transient Data**. Because **Flipbook Build Mode** is on, the **Sweep** button on the **Transient Data Controls** panel is replaced by **Build**. Click **Build**.
5. The **Streakline Export** panel will prompt to save the streaklines (based on our visible streamline rake seed locations) to a Particle Path file, which makes subsequent streakline display much simpler. Click **Yes**. In the **Export Streaklines** panel (not shown), name and **Save** the .fvp file. [\[Page 200 Working with FieldView\]](#)
6. Once the 51 frames are created, the **Flipbook Controls** panel can be used to play and **Save** the animation. [\[Page 309 Reference Manual\]](#)

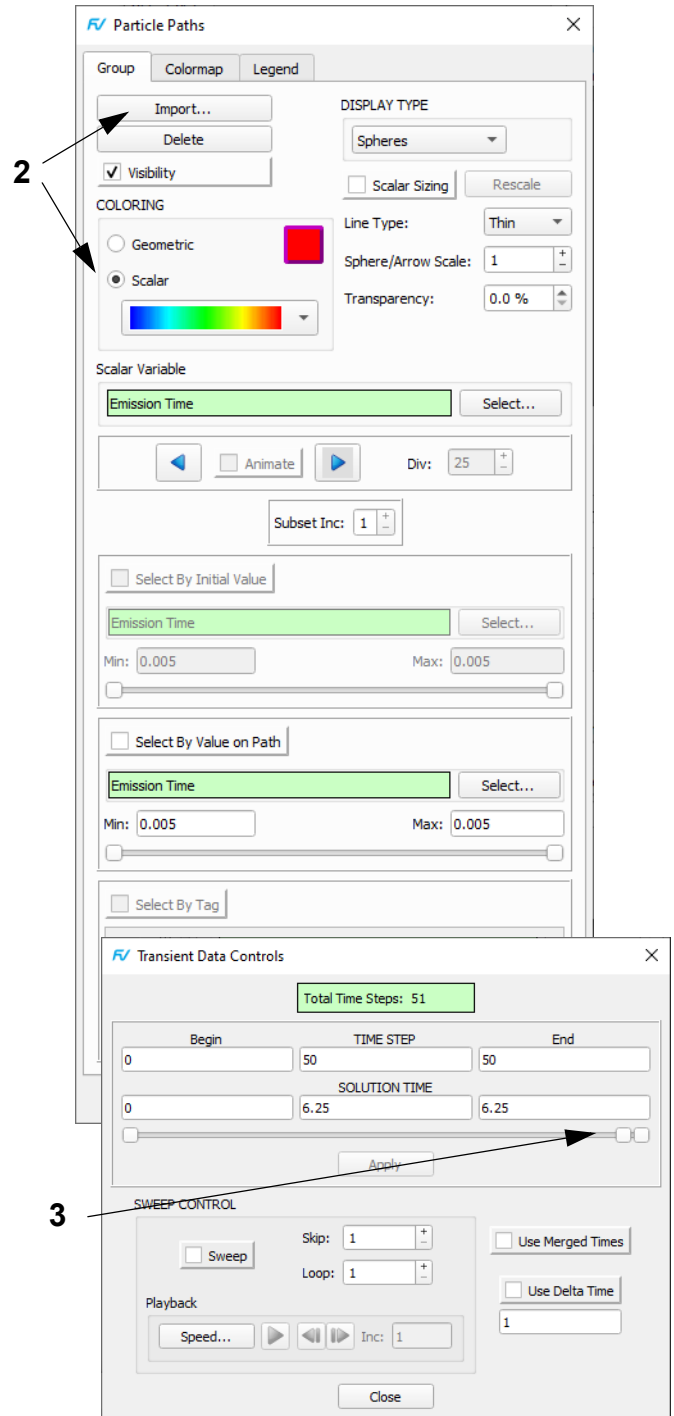


Step 6: Import Streaklines and Improve your Animation

In this step, we import our previously exported streaklines. We will then create another animation which will allow us to better see the vortices being shed off of the cylinder.

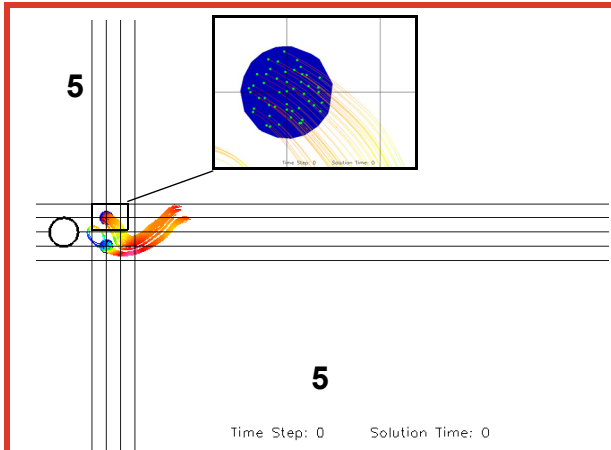


1. On the **Streamlines** panel (not shown), turn off the **Visibility** for each rake. The arrow buttons at the top of the panel can be used to switch between Rake ID 1 and Rake ID 2.
2. Bring up the **Particle Paths** panel. Press the **Import** button. Browse to and select one of the many streakline export (particle set) files created previously. Change the **COLORING** to **Scalar** and **DISPLAY TYPE** to **Dots**. Click **Select...** and choose **Emission Time** as the **Scalar Variable** for the particles. [\[Page 215 Working with FieldView\]](#)
3. Bring up the **Transient Data Controls** panel. Advance the **TIME STEP** to 50 and click **Apply**. This will show the positions of the particles at their furthest extent. When the **TIME STEP** slider is moved and applied, the positions of the particles change to match their locations at the selected time step.
4. Set **Object:** to **World** on the Transform Controls toolbar. Resize the image in the graphics window so that more can be seen. Use zooming out (M3) and left translation (M1).
5. Bring up the **Coordinate Surface** panel. Turn off the **Visibility** of Surfaces #11 and #12 and turn on the **Visibility** of Surface #1 (the contour surface). On the **Transient Controls** panel, move the slider back to **TIME STEP 0** and press **Apply**.
6. Turn **Flipbook Build Mode** on (**Tools** → **Flipbook Build Mode**). Press the **Build** button on the **Transient Data Controls** panel to create a flipbook. **Save** the flipbook when creation is complete (actions not shown).



Step 7: Using Scripts and DataGuide™

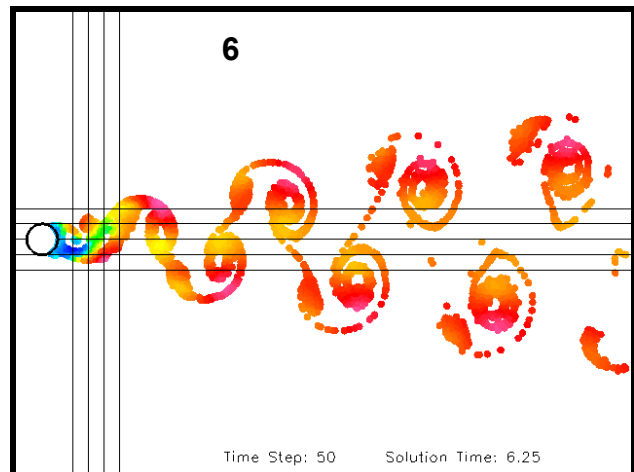
In this section, we will build a flipbook and produce a streakline export file at the same time using a **FieldView** Script file. We will then import the exported streaklines and sweep through the dataset again, creating a flipbook of the imported Particle Paths. This tutorial can also be used in conjunction with **DataGuide™**.



export file, **100_seeds.fvp** (a binary Particle Path file) will be produced along with the flipbook **simple_streak.mp4** (for LINUX) or **simple_streak.avi** (for WINDOWS). You should see something similar to the illustration shown below at the end of the transient sweep.

```
!..FieldView Script:
!   Start recording a flipbook
RECORD ON simple_streak
!   Performs a transient sweep &
!   saves a streakline file
SWEEP TIME LOOP 1 1 1 100_seeds.fvp
!   Turn off recording
RECORD OFF
!..End of Script
```

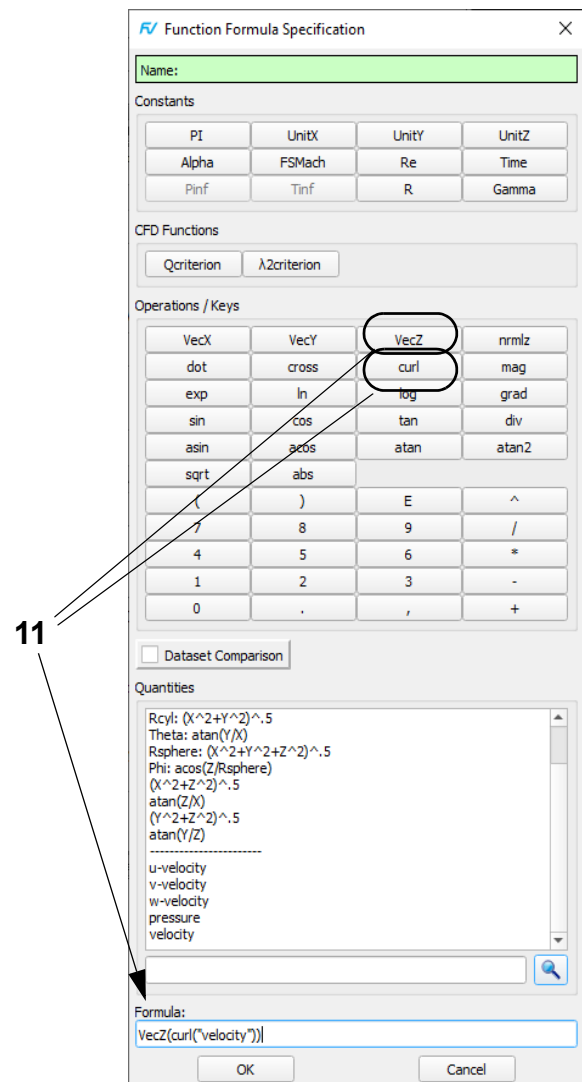
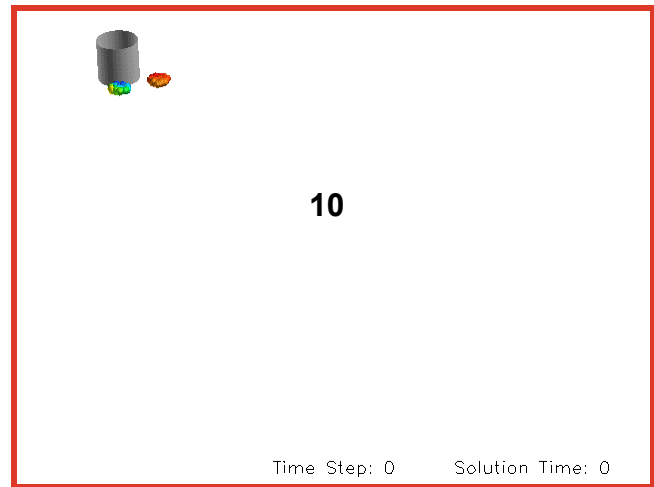
- Copy the tutorial data, scripts and restarts (found in the **/vortexshedding** sub-directory of the **/tutorial** directory on the **FieldView** DVD-ROM) to your local hard disk. You will need write permission to this directory since files will be created. Approximate file space needed will be: 52 MB (no **DataGuide™**) or 68 MB (with **DataGuide™**).
- Start **FieldView** with the **DataGuide™** switch (add the **"-p2"** command-line switch to your usual **FieldView** start-up command). Consult the **DataGuide™** section [Page 9 User's Guide] for information about how to do this.
- Use the **Tools→Graphics Layout Size** pull-down and choose **640x480 NTSC**.
- From **File → Open Restart → Complete...** read in the Complete Restart called **setup_streakline.dat**. This restart recreates the above view (minus the inset image), which is similar to that produced by completing the first 6 steps of this tutorial but using 50 seeds instead of 10. The inset image is the view after zooming into the seeded surface to show the position of the 50 seeds.
- From **File → Open Restart → Script...** read the Script restart called **create_streakline.scr**. A streakline
- Note that for each time step, a 327 kb **DataGuide™** Grid Processing file (*.fvpre) and a 1 kb **DataGuide™** Results Processing file (*.fvres) are created.
- Exit from **FieldView**. Make sure that the file **100_seeds.fvp** and the flipbook were created.
- Start **FieldView** again, this time without using any **DataGuide™** command-line switches.

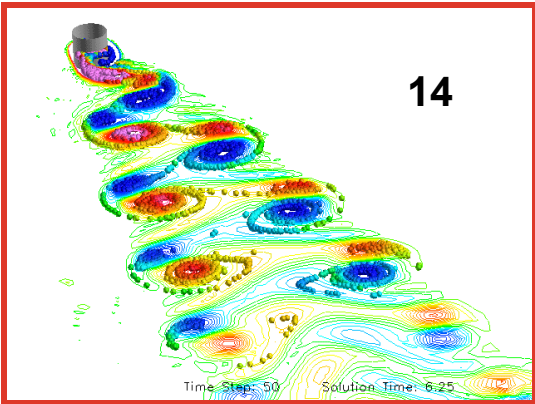
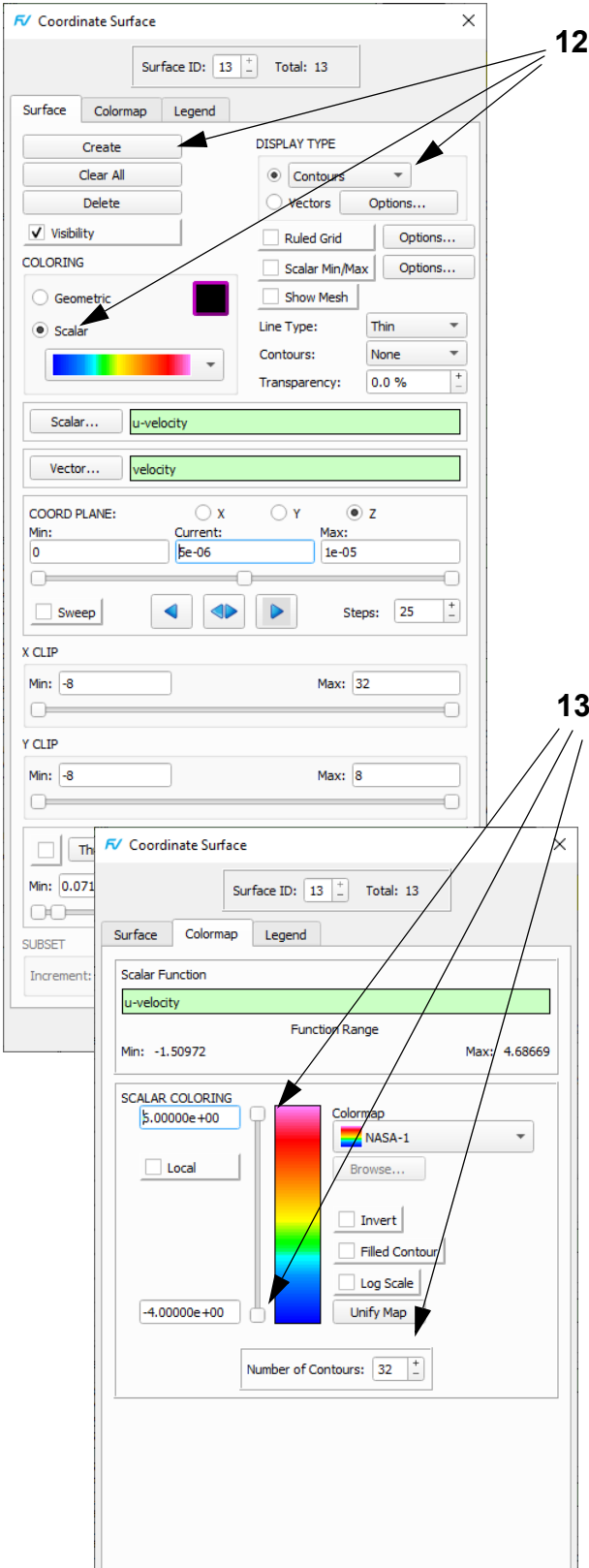


9. From **File** → **Open Restart** → **Complete...** read in the Complete Restart called **display_streakline.dat**. This will recreate an image similar to that shown at right. This restart imports the streaklines as Particle Paths, removes the coordinate lines, draws the cylinder using Smooth shading, changes the view to better display the vortex shedding, and turns on Presentation Rendering for better looking particles. In addition, the dataset restart scales the dataset with a Z-Scale factor of 100,000 to make the cylinder more visible.
10. Click the main toolbar icon OR **Edit** menu to bring up the **Function Specification** panel and **Create**. On the **Function Formula Specification** panel, create the **Formula: VecZ(curl(velocity))**; click **OK** and, on the **Function Name Input** panel, **Use Formula**. On the **Function Specification** panel, click **Scalar** and, on the **Function Selection** panel, select **VecZ(curl(velocity))** and click **Calculate** to load the new function into the **Scalar** register.
11. **Create** a Z Coordinate Surface (at the default location of $Z=5e-06$). Change the **COLORING** from **Geometric** to **Scalar** and the **DISPLAY TYPE** to **Contours**, and turn off **Threshold Clip**.
12. Press the **Colormap** tab on the **Coordinate Surface** panel. Change the scalar min/max values to **-4** and **6**, respectively. The **Number of Contours** should be **32**. Alternately, these changes can be made using the **Scalar Colormap Specification** panel.
13. From **File** → **Open Restart** → **Script...** read in the second Script restart called **display_streakline.scr**.

```
!..FieldView Script which:
!   Starts recording a flipbook
RECORD ON final_streak
!   Performs a transient sweep
!   (no streakline file saved)
SWEEP TIME LOOP 1 1 1
!   Turns off recording
RECORD OFF
!..End of Script
```

14. Exit **FieldView** (if desired) and play the two flipbook animations.





Chapter 9

9

Polymer Processing Tutorial

What You Will Learn

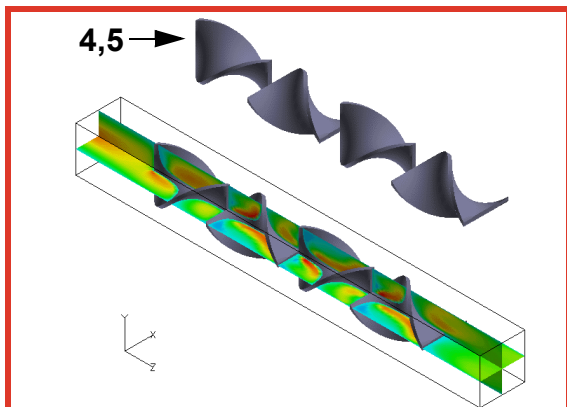
- How to visualize back flow on mixer blades
- How to create multiple cross sections along side the mixer to visualize axial velocities
- How to increase colormap contours
- Create a legend and change the number of decimal places
- How to display shear rates on blade surfaces
- How to integrate the concentration variance on a surface
- How to import a Restart File
- How to translate surfaces and create a titled “fade-in” for a Keyframe Animation

Step 1: Create Boundary Surfaces

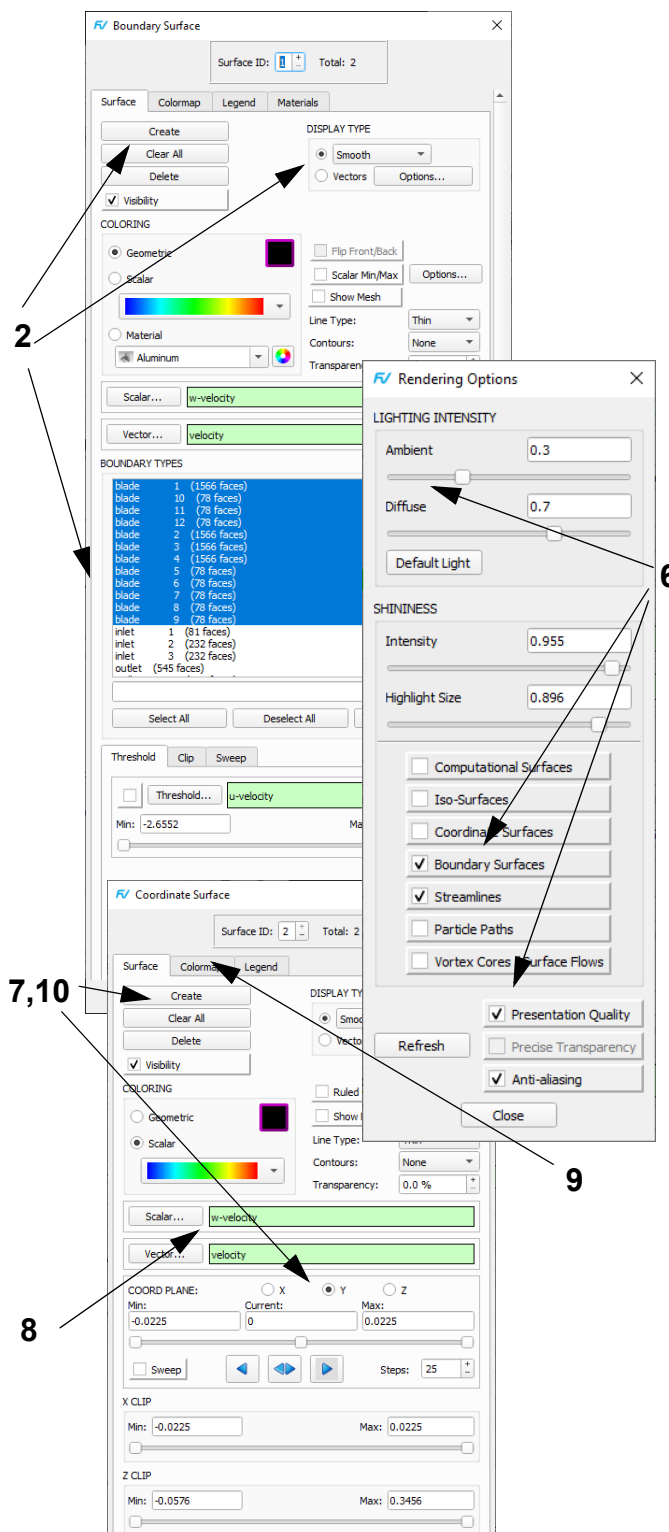
This tutorial shows how to create advanced functions in **FieldView** to help quantify the extent of mixing in a static mixer.



The pictures on each page correspond to the final image that should be created if the steps are completed correctly.

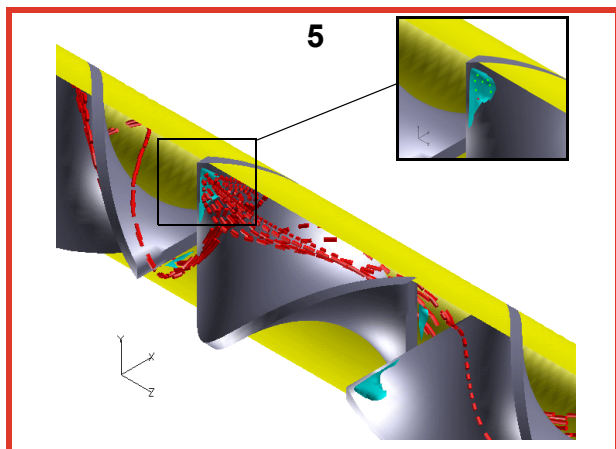


1. Bring up the **File** → **Data Input** → **AcuSolve [FV-UNS Export]** panel. Click **Read Grid or Combined Data...** In the file browser, select **kenics.uns** and **Open**. **Close** the **AcuSolve [FV-UNS Export]** panel. [Page 36 Reference Manual]
2. Bring up the **Boundary Surface** panel. Click **Create**, select **blade 1, blade 2, ... , blade 12** to create a single surface consisting of all of the blades and click **OK**. Change **DISPLAY TYPE** to **Smooth** shading.
3. Bring up the **Tools** → **Color Mixer** panel. Change the gray chip to **Red, Green and Blue** values of **130, 130 and 158**. **Apply** and **Close**. In the **Boundary Surface** panel, **BOUNDARY TYPES** section, click **OK**.
4. Click **Create**, select all 12 blades again and click **OK**.
5. On the Transform Controls toolbar, change **Object:** to **Surface**. Use the left mouse (M1) to translate the current boundary surface up and right. Click **Viewer Options...**, turn **Perspective** off and **Close**.
6. Bring up the **View** → **Rendering Options** panel. Turn off everything except **Boundary Surfaces** and **Streamlines**. Turn on **Presentation Quality**. Adjust the **Intensity** and **Highlight Size**, click **Refresh** and **Close**. [Page 360 Working with FieldView]
7. **Create** an **X Coordinate Surface**. Change **DISPLAY TYPE** to **Smooth** shading and **COLORING** to **Scalar**.
8. Change the **Scalar** function to **w-velocity**.
9. Change the **Colormap** maximum value to **2**.
10. **Create** a second coordinate surface at **Y = 0**.

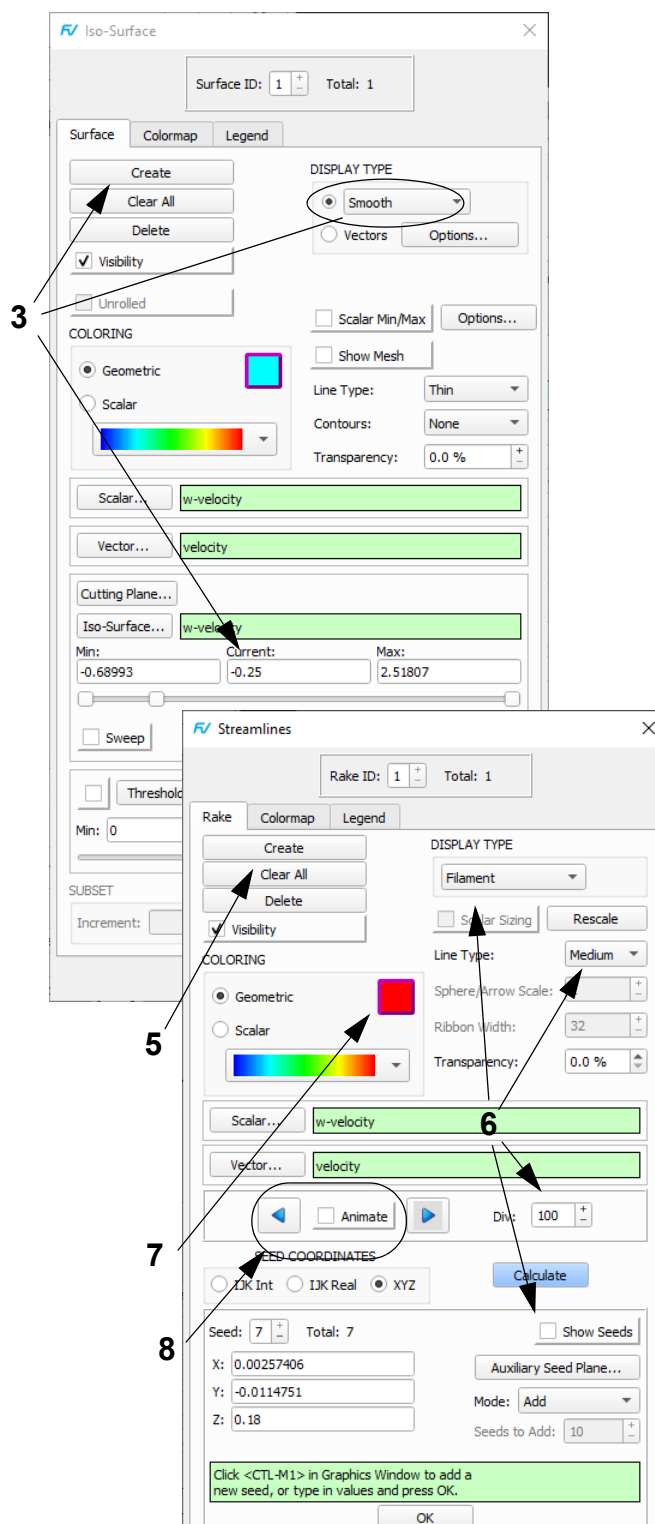


Step 2: Visualize Back Flow

This step will help visualize the back flow resulting from the mixer blades.

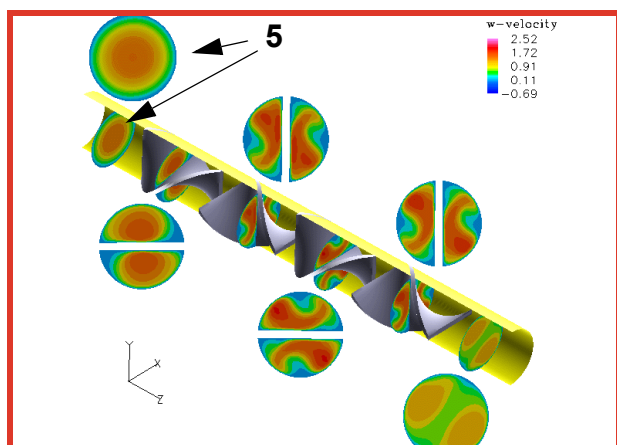


1. On the Transform Controls toolbar, turn Outline off. Turn off the **Visibility** of the coordinate surfaces and the surface transformed boundary surface. On the Viewer toolbar, set **Object:** to **World**. On the side toolbar, click the **Zoom** icon and use **M1** (left mouse) to select an area to zoom into the third blade element.
2. **Create** a single boundary surface consisting of **wall 1**, ... , **wall 10**. Turn on **Visibility** (if not already). Change the **Geometric COLORING** to yellow. On the **Function Specification** panel, change the **Threshold** function to **X**. On the **Boundary Surface** panel, turn on **Threshold Clip**, set the **Min:** to **0** and Enter.
3. Bring up the **Iso-Surface** panel. Click **Create** and **Define Iso-Function...** Select **w-velocity** and **Calculate**. Change the **Current:** value to **-0.25** and the **DISPLAY TYPE** from **Constant** to **Smooth** shading. We will seed streamlines on this iso-surface. [Page 170 Working with FieldView]
4. Change the **Geometric COLORING** to light blue.
5. Bring up the **Streamlines** panel. On the **Rake** tab, click **Create**. **Zoom** with **M1** further into the largest iso-surface region for easier seeding. Use **CTRL-M1** to add **8** seeds. In the **Calculation Parameters** section, change **Direction** to **Both** and the **Step** value to **6**. Click **Calculate**. [Page 184 Working with FieldView]
6. Change the **DISPLAY TYPE** to **Filament**, **Line Type** to **Medium**, and **Div:** to **100**. Turn **Show Seeds** off. Use **M3** (right mouse) to undo the zoom.
7. Click on the colorchip, and use the **Colormap Specification** panel (not shown), to change **Geometric** to red.
8. Turn on **Animate**. Turn off **Animate** for the next step.

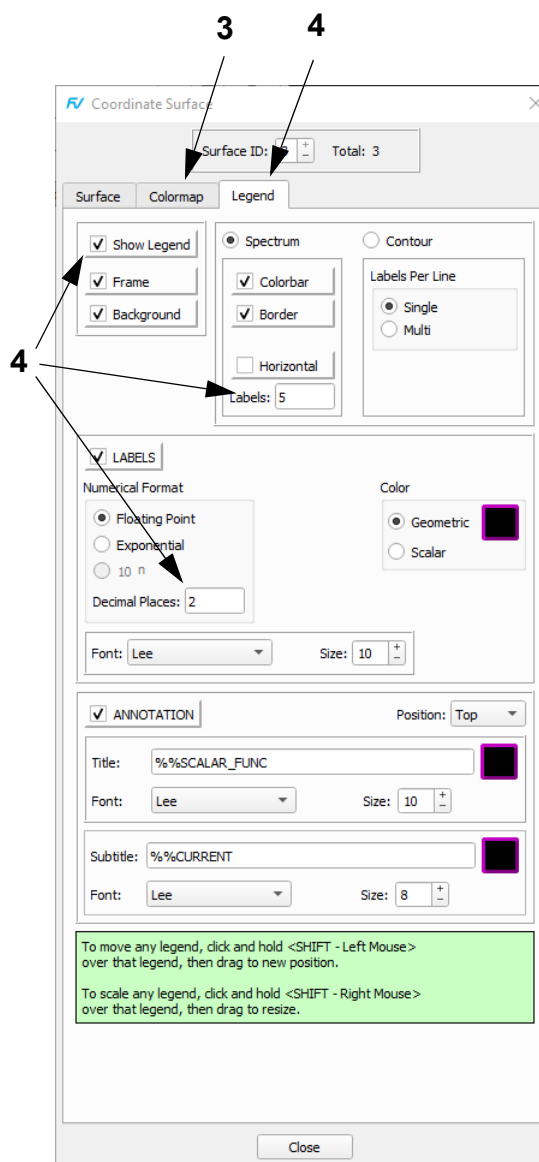


Step 3: View Axial Velocities

We will sample cross-sections of the static mixer to visualize axial velocities. A restart is available to recreate this view.



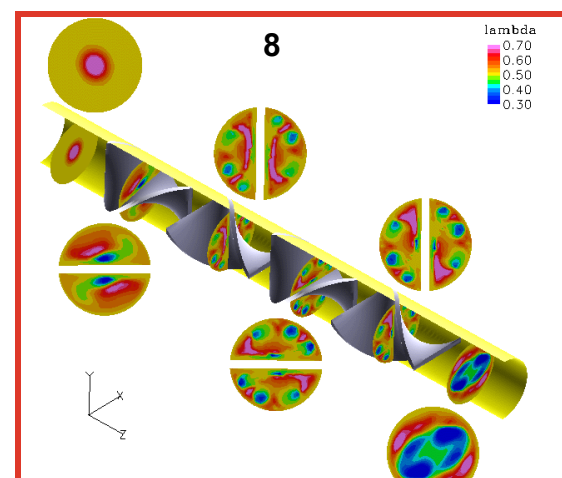
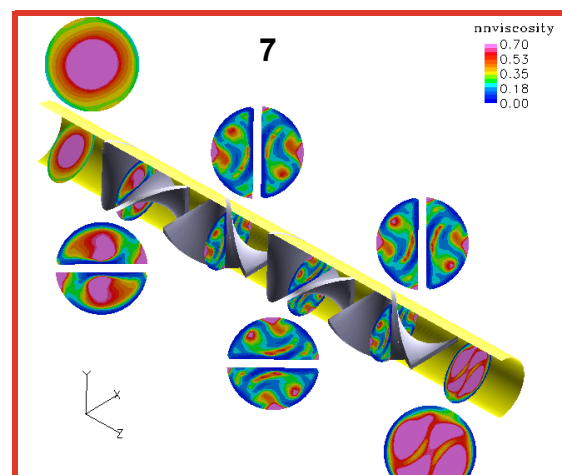
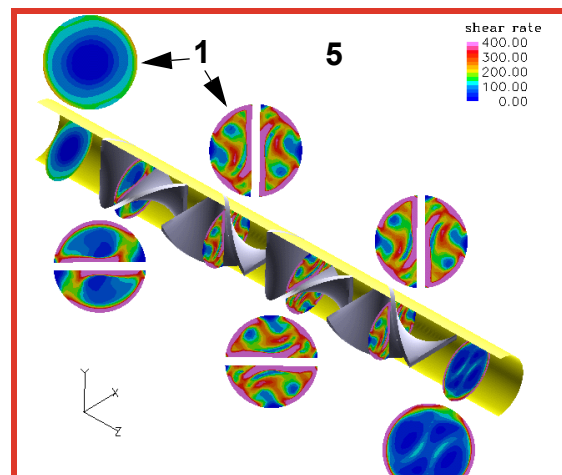
1. Turn off the **Visibility** of the streamlines and the iso-surface. Zoom out to view the entire dataset.
2. **Create** a coordinate surface at **Z = -0.0288**. Turn on **Visibility** (if needed). The **Scalar Function** should still be **w-velocity**, the **DISPLAY TYPE** **Smooth** shading and the **COLORING** **Scalar**. [\[Page 232 Working with FieldView\]](#)
3. On the **Colormap** tab, change the colormap from **Spectrum** to **NASA-1**. Turn on **Filled Contour**. Change the **Number of Contours** to **64**.
4. On the **Legend** tab, turn **Show Legend** on, change **Decimal Places** to 2 and the number of **Labels** to 5. Hold the Shift key down, position the mouse over the legend, then left click and drag to position the legend.
5. Click **Create** again to create an identical surface. This one will be extracted.
6. On the Viewer toolbar, set **Object:** to **Surface**. Use the mouse to move, rotate and zoom the surface as illustrated.
7. Bring up **Visualization Panels** → **Annotation...** Click **Create Text**, type **w-velocity** and click **OK**. Use Shift + **M1** (left mouse) to reposition the title (panel not shown). [\[Page 224 Working with FieldView\]](#)
8. Repeat steps 2, 5 and 6 five more times, creating surfaces at **Z** values of **0.0306**, **0.108**, **0.18**, **0.252** and **0.3168**, or use **File.. Open Restart.. Coordinate Surface..** to read in the Coordinate Surface restart called **polymer.crd** in the polymer processing directory to obtain the image seen above.



Step 4: Change the Scalar Display of the Cross Sections

This step will show how to create similar images of several other useful flow properties. Defining these flow properties involves using the function calculator to define the equations and variables needed. A listing of the formulae used follows.

1. "Quick-pick" (double-click) on any one of the coordinate surfaces. [\[Page 75 User's Guide\]](#)
2. The formulae needed can be found in a Formula Restart. Bring up the **File** → **Open Restart** → **Formula...** browser. In the polymer_processing directory, select **polymer.frm** and click **Open**. [\[Page 260 Reference Manual\]](#)
3. Click the **Functions** icon on the main toolbar. Click **Scalar...** On the **Function Selection** panel, select **shear** and click **Calculate**. Close the **Function Specification** panel.
4. On the **Colormap Specification** panel, change the minimum and maximum values to **0.0** and **400**.
5. Steps 2-4 above display the function **shear** on the current surface. To force all the coordinate surfaces in the dataset to acquire the same attributes as the current surface, use **Tools** → **Unify**. This will also update the Legend, since it belongs to one of the other coordinate surfaces. **Unify** applies changes to all surfaces of the same type only.
6. "Quick-pick" the title. On the **Annotation** panel, **Edit Text** from **w-velocity** to **shear rate** and Enter. Note that the highest shear rates occur at the blade and wall surfaces. This is important because it shows where product damage is most likely to occur and may be very critical for sensitive materials.
7. Quick pick a coordinate surface. Repeat step 3 for **nnviscosity** (non-Newtonian viscosity). Change the colormap range to **0.001** to **0.700**. **Unify**. On the **Annotation** panel, **Edit Text** to **nnviscosity**. This image complements the shear rate picture. Areas of high shear tend to be associated with areas of lower viscosity. This is a good qualitative check. The significance for mixing is that if there are regions of high viscosity, the mixing there is likely to be low.
8. Quick pick a coordinate surface. Repeat step 3 for the mixing parameter, **lambda**. Adjust the range to **0.30** to **0.70**. **Unify**. On the **Annotation** panel, **Edit Text** to **lambda**. The mixing parameter is interpreted as follows:
 0.0= simple rotation
 0.5=simple shear
 1.0=elongation
 The blue areas are bad because the fluid there is essentially turning as a solid body. The pink areas are bad because the fluid there is being stretched.



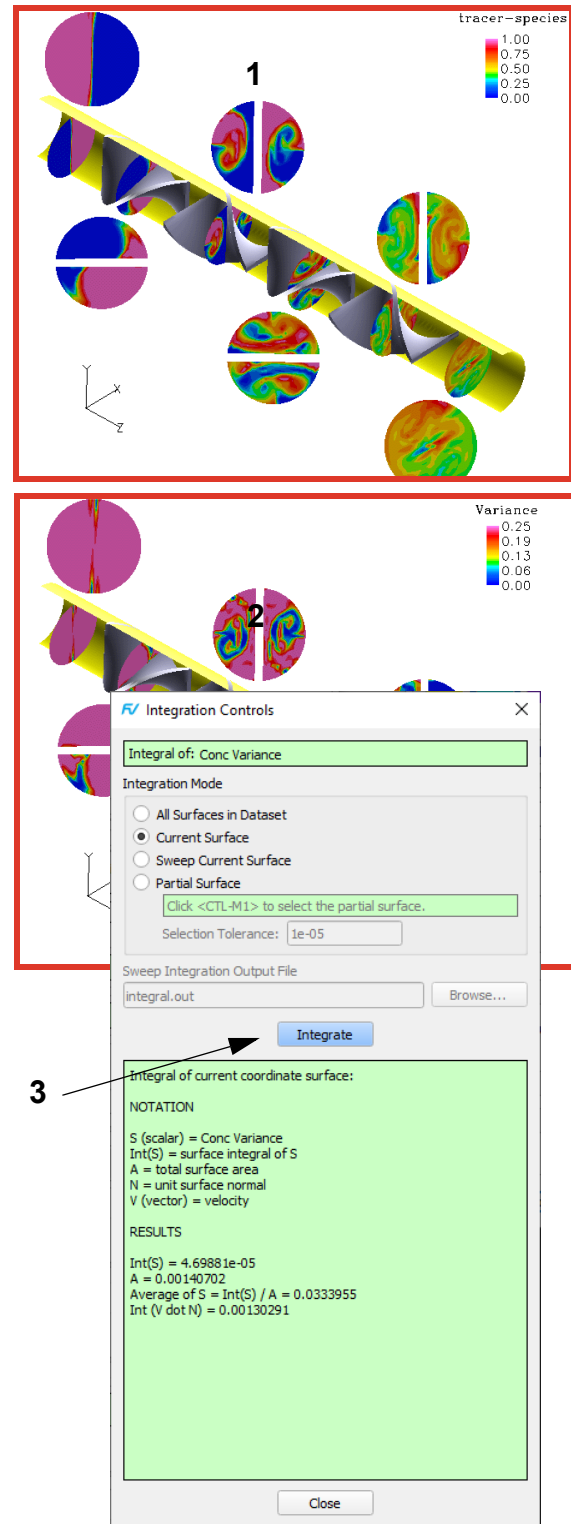
Step 5: Integrate Concentration Variance of Each Cross Section

This step will be similar to the last step in that we will change the **Scalar Function** being displayed on the coordinate surface.

1. Quick pick a coordinate surface. Click the **Functions** icon on the main toolbar. Click **Scalar...** On the **Function Selection** panel, select **species conc 1** and click **Calculate**. Close the **Function Specification** panel. Adjust the range to **0.00** to **1.00**. **Unify**. On the **Annotation** panel, **Edit Text** to **tracer-species**. A tracer species can be used to show mixing patterns. By setting up one-half of the inlet with a concentration equal to 1.0, and the other half equal to zero, you can see how fluid from each part of the inlet mixes together. Note that the extent of mixing differs significantly at each cross-section.
2. Quick pick a coordinate surface. Click the **Functions** icon on the main toolbar. Click **Scalar...** On the **Function Selection** panel, select **Conc variance** and click **Calculate**. Close the **Function Specification** panel. Adjust the range to **0.00** to **0.25**. **Unify**. On the **Annotation** panel, **Edit Text** to **Variance**. Species variance shows the extent of mixing statistically. At the inlet, the fluid is completely unmixed, and in this case, the variance should be equal to **0.25** (Note: average concentration throughout the mixer will always be 0.5). At each of the cross-sections, the variance decreases:

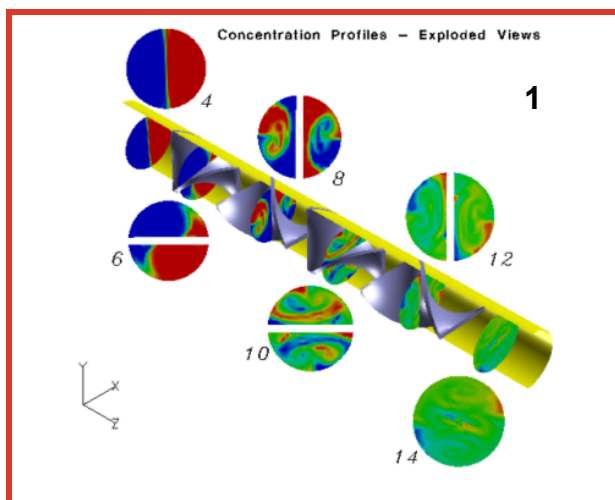
Section #	Variance (avg.)
1	0.2448
2	0.2126
3	0.1770
4	0.0760
5	0.0334
6	0.0190

3. To compute these numbers, double-click on each of the six sections to make it current. Click the **Integrate** icon on the side toolbar OR navigate to **Tools** → **Integration...** Change the **Integration Mode** to **Current Surface**. Click **Integrate** to calculate the integral of the scalar function over the surface. Close the panel. Repeat for the other surfaces. Panel image at right shows **RESULTS** for Section #5.

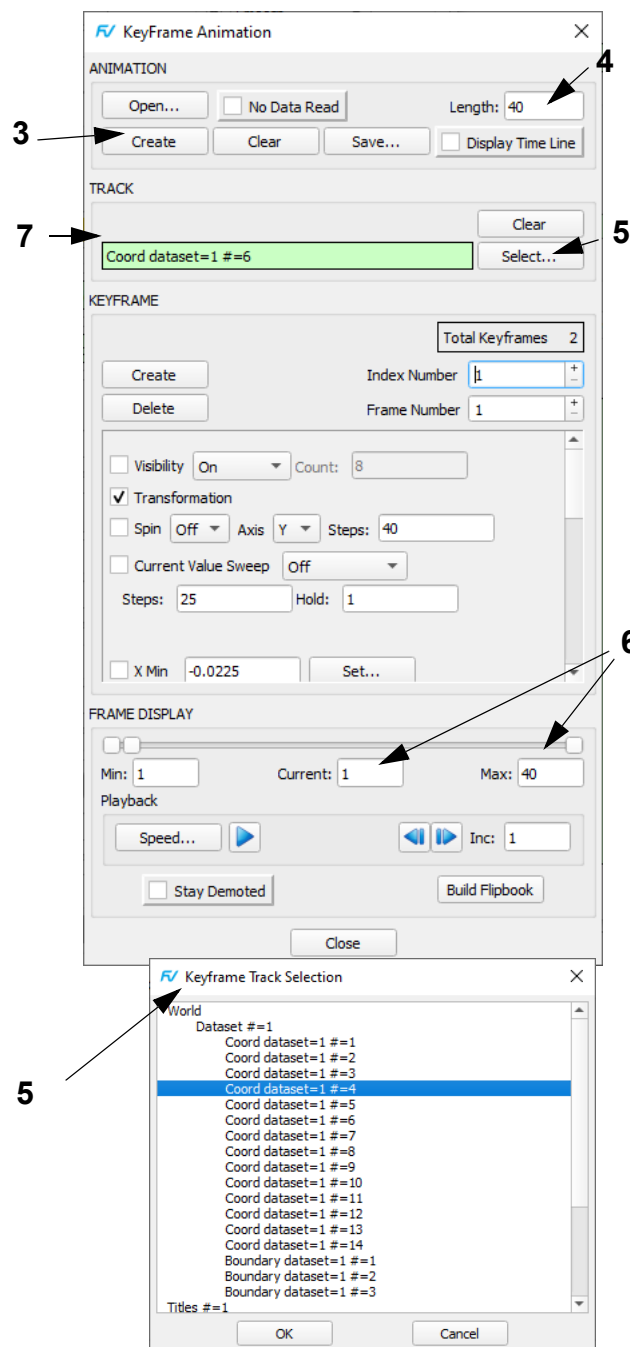


Step 6: Create a Keyframe Animation

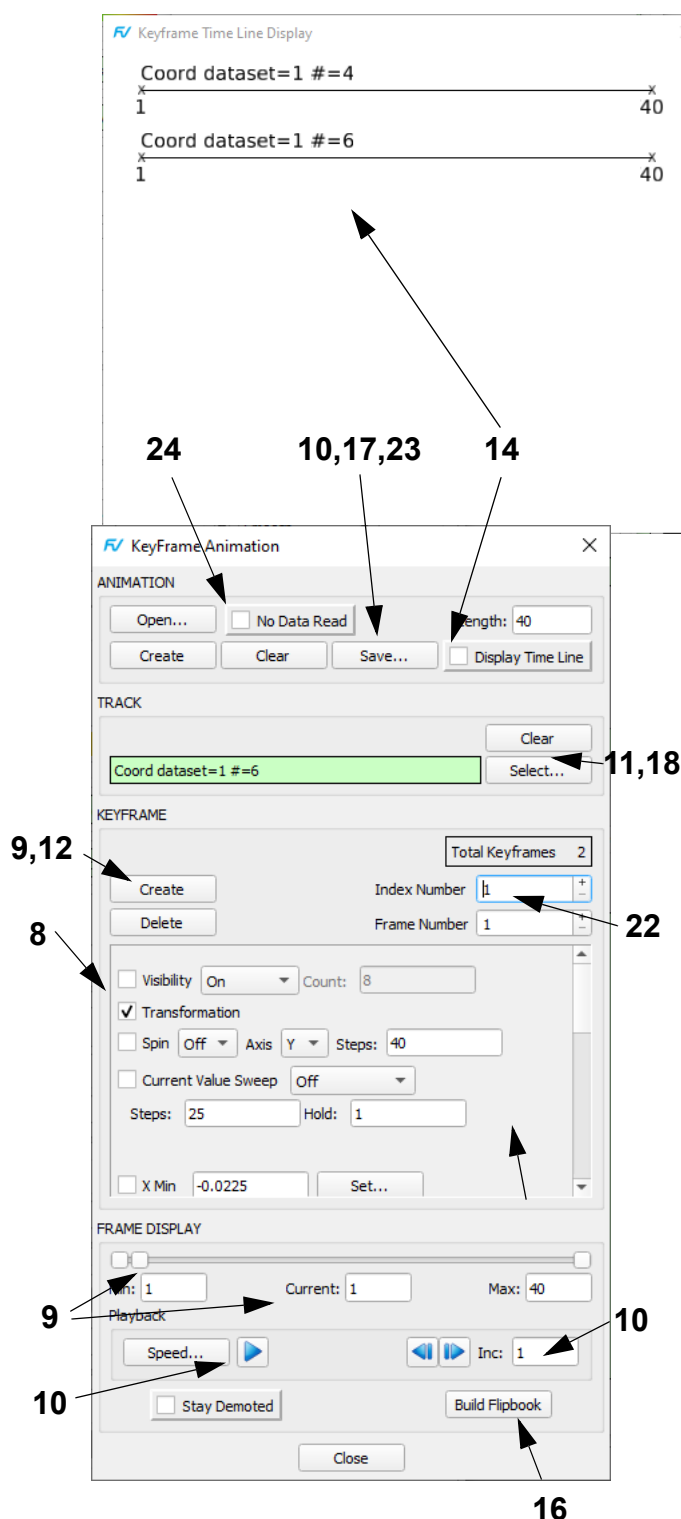
This section will show how to create a keyframe animation of an exploded view. This section uses a Complete Restart and a Keyframe Restart which accompany the tutorial data. Keyframe animations require some planning before execution, due to the complexity of the animation that can be created.



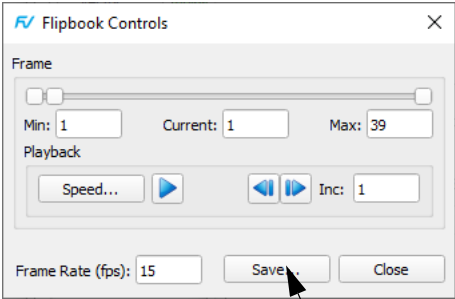
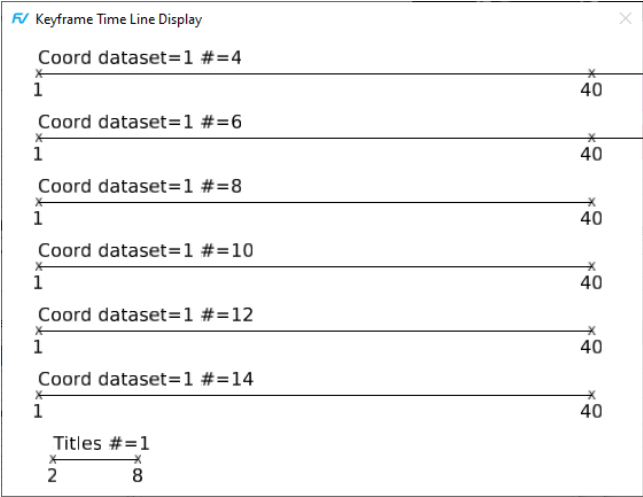
1. Copy the data and restarts from the `/tutorial/polymer_processing` sub-directory on the **FieldView** DVD-ROM to your local hard disk. Bring up the **File** → **Open Restart** → **Complete...** browser. In the `polymer_processing` directory, select **stat_key-start.dat** and click **Open**. This restart creates an image similar to that shown at the top of the previous page, except that the legend has been removed, the title has been changed from **tracer-species** to **Concentration Profiles - Exploded Views** and exploded coordinate surface numbers have been added for easier identification.
2. From **Tools** → **Graphics Layout Size**, choose **640x480 NTSC**. The specified size will be used when creating images or animations.
3. Bring up the **Tools** → **Keyframe Animation...** panel. In the **ANIMATION** section, click **Create** to create a new keyframe animation. [\[Page 316 Reference Manual\]](#)
4. In the **Length:** field, double-click **120**, change the default keyframe animation length to **40**, and hit Enter or Return. The animation will not need to be longer and this will also help display the time line more clearly.
5. Click **Select...** In the **Keyframe Track Selection** panel, select **Coord dataset=1 #=4** and click **OK**.



6. In the **FRAME DISPLAY** section, move the slider to **40** OR edit **Current:** (double-click in the field, type **40** and Enter). (See the panel on the previous page).
7. In the **KEYFRAME** section, click **Create** to create a keyframe at frame **40** for **Coord dataset=1 #=4**.
8. Turn on **Transformation** to set the position of this particular surface for keyframe number **1**, which corresponds to a frame number of **40**. This is the final position of the surface in the animation.
9. To set the initial position of the surface, set the **Current:** frame to **1**. In the **KEYFRAME** section, click **Create**. On the Transform Controls toolbar, click **Reset**. This will place the surface at its original, untransformed position and turn on **Transformation**.
10. Click the play arrow to preview your animation or the **Frame Advance** arrows to preview specific frames. To speed up the preview, edit the **Inc:** value to 2 or 4 to play only every other or every 4th frame. **Save** a keyframe restart.
11. Click **Select...**, select **Coord dataset=1 #=6** and click **OK** to set the two keyframes for the next coordinate surface.
12. Change the **Current:** frame to **40**. In the **KEYFRAME** section, click **Create** and turn on **Transformation**.
13. Change the **Current:** frame to **1**. In the **KEYFRAME** section, click **Create**. On the Transform Controls toolbar, click **Reset** to set the initial position of the surface.
14. Turn on **Display Time Line**. In the **Keyframe Time-line Track Selection** panel, click **OK** to bring up the **Keyframe Time Line Display** panel for the selected tracks. The **Keyframe Time Line Display** panel can be manually resized. Turn off **Display Time Line** to dismiss the panel.
15. Repeat steps 11-13 for coordinate surface numbers 8, 10, 12 and 14.
16. Click **Build Flipbook** to create a keyframe animation file. In the **Flipbook Display Size Warning** panel, click **OK**. In the **Flipbook Controls** panel, click **Save...** In the **Flipbook File Save** browser, **Save** the file. **Close** the **Flipbook Controls** panel.
17. In the **KeyFrame Animation** panel, **Save** a keyframe restart again.



- 18. The title will fade in and move from near the mixer to its current position, but faster than the surfaces. Click **Select...**, select **Titles #=1** and click **OK**.
- 19. The title's current position will be its final position. Change the **Current:** frame to **8**. In the **KEYFRAME** section, click **Create** and turn on **Transformation**.
- 20. Change the **Current:** frame to **1**. In the **KEYFRAME** section, click **Create**. Change the **Visibility** pull-down from **On** to **Fade In**. Note that the fade-in has already started; the fade starts immediately on frame **1**, which has a fade (i.e., transparency) value of **0.125**.
- 21. With Shift + left mouse, move the title closer to the mixer. This will automatically turn on the **Transformation** button and set the keyframe.
- 22. To have the title start invisible, the fade-in needs to start on frame #2 instead of frame #1. In the **Frame Number** field, click the plus sign (+) to change the position (of the title keyframe only) to **2**.
- 23. The final Time Line is shown to the right. Click **Build Flipbook** to create a new keyframe animation file and **Save**. **Save** the final keyframe restart.
- 24. A completed keyframe restart (**stat_anim.key**) has been provided and can be opened if desired. This restart contains all of the keyframes and actions that are produced by the above steps.



23

Equations for Static Mixer

A number of mathematical expressions were used to create the visualizations presented in this tutorial and have been provided as Formula Restarts. An explanation of the equations found in the formula restarts and used in the tutorial is given here.

The following definitions were used in the static mixer equations:

Non-Newtonian Viscosity	$\eta = \frac{\tau_o}{\ \Delta\ } + \eta_o \cdot e^{(-t_1 \ \Delta\)}$ $\tau_o = 41.7$ $\eta_o = 0.0115$ $t_1 = 4.52 \times 10^{-5}$	τ_o , η_o and t_1 are model parameters η = non-newtonian viscosity Δ = rate of deformation tensor
Rate of Deformation Tensor	$\Delta_{ij} = \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right)$	
Shear Rate Magnitude	$\ \Delta\ = \sqrt{\frac{1}{2} \cdot (\Delta : \Delta)}$ $(\Delta : \Delta) = 4 \left[\left(\frac{\partial v_x}{\partial x} \right)^2 + \left(\frac{\partial v_y}{\partial y} \right)^2 + \left(\frac{\partial v_z}{\partial z} \right)^2 \right] + 2 \left[\left(\frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right)^2 + \left(\frac{\partial v_x}{\partial z} + \frac{\partial v_z}{\partial x} \right)^2 + \left(\frac{\partial v_y}{\partial z} + \frac{\partial v_z}{\partial y} \right)^2 \right]$	
Shear Stress	$\tau = \eta \cdot \ \Delta\ $	
Vorticity	$\omega = [\nabla \times V]$	
Mixing Parameter	$\lambda = \frac{\ \Delta\ }{\ \Delta\ + \omega }$	

FieldView Equations

In the following equations, the name of the function as stored in the restart files and as appears in **FieldView** when read in is shown first, followed by the mathematical expression followed by the expression used in **FieldView** to define the given function. All of the terms and factors of the expressions used in **FieldView** are either intrinsic functions available on the Function Formula Specification panel or have been previously defined in this section.

$$\begin{aligned}
 \text{Dxx} &= \frac{\partial v_x}{\partial x} = \text{VecX}(\text{grad}(\text{"u - velocity"})) & \text{Dxy} &= \frac{\partial v_x}{\partial y} = \text{VecY}(\text{grad}(\text{"u - velocity"})) & \text{Dxz} &= \frac{\partial v_x}{\partial z} = \text{VecZ}(\text{grad}(\text{"u - velocity"})) \\
 \text{Dyx} &= \frac{\partial v_y}{\partial x} = \text{VecX}(\text{grad}(\text{"v - velocity"})) & \text{Dyy} &= \frac{\partial v_y}{\partial y} = \text{VecY}(\text{grad}(\text{"v - velocity"})) & \text{Dyz} &= \frac{\partial v_y}{\partial z} = \text{VecZ}(\text{grad}(\text{"v - velocity"})) \\
 \text{Dzx} &= \frac{\partial v_z}{\partial x} = \text{VecX}(\text{grad}(\text{"w - velocity"})) & \text{Dzy} &= \frac{\partial v_z}{\partial y} = \text{VecY}(\text{grad}(\text{"w - velocity"})) & \text{Dzz} &= \frac{\partial v_z}{\partial z} = \text{VecZ}(\text{grad}(\text{"w - velocity"}))
 \end{aligned}$$

etc. for Dxy, Dyy, Dzy and Dxz, Dyx, and Dzz.

$$\begin{aligned} \text{DELxx} &= \left(\frac{\partial v_x}{\partial x} \right)^2 = \text{"Dxx"}^2 & \text{DELxy} &= \left(\frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right)^2 = (\text{"Dxy"} + \text{"Dyx"})^2 \\ \text{DELYy} &= \left(\frac{\partial v_y}{\partial y} \right)^2 = \text{"Dyy"}^2 & \text{DELxz} &= \left(\frac{\partial v_x}{\partial z} + \frac{\partial v_z}{\partial x} \right)^2 = (\text{"Dxz"} + \text{"Dzx"})^2 \\ \text{DELzz} &= \left(\frac{\partial v_z}{\partial z} \right)^2 = \text{"Dzz"}^2 & \text{DELYz} &= \left(\frac{\partial v_y}{\partial z} + \frac{\partial v_z}{\partial y} \right)^2 = (\text{"Dyz"} + \text{"Dzy"})^2 \end{aligned}$$

$$\mathbf{D} : \mathbf{D} = \mathbf{\Delta} : \mathbf{\Delta} = 4(\text{"DELxx"} + \text{"DELYy"} + \text{"DELzz"}) + 2(\text{"DELxy"} + \text{"DELxz"} + \text{"DELYz"})$$

$$\begin{aligned} \text{shear} &= \sqrt{\frac{1}{2} \cdot \mathbf{\Delta} : \mathbf{\Delta}} = \text{sqrt}(0.5 \cdot \mathbf{D} : \mathbf{D}) \\ \text{nnviscosity} &= \frac{\tau_o}{\|\mathbf{\Delta}\|} + \eta_o \cdot e^{(-\tau_i \|\mathbf{\Delta}\|)} = \frac{41.7}{(\text{"shear"} + \varepsilon)} + 0.0115 \times \exp(-4.52\text{E} - 05 \times \text{"shear"}) \\ \text{stress} &= \eta \cdot \|\mathbf{\Delta}\| = \text{"nnviscosity"} \times \text{"shear"} \\ \text{vorticity} &= |\nabla \times \mathbf{V}| = \text{mag}(\text{curl}(\text{"velocity"})) \\ \text{lambda} &= \frac{\text{"shear"} + 0.5 \cdot \varepsilon}{\text{"shear"} + \text{"vorticity"} + 1.0 \cdot \varepsilon}, \varepsilon = 1.0 \times 10^{-6} \end{aligned}$$

Note: Use ε to avoid division by zero problems.

Chapter 10

10

Biomedical Tutorial

What You Will Learn

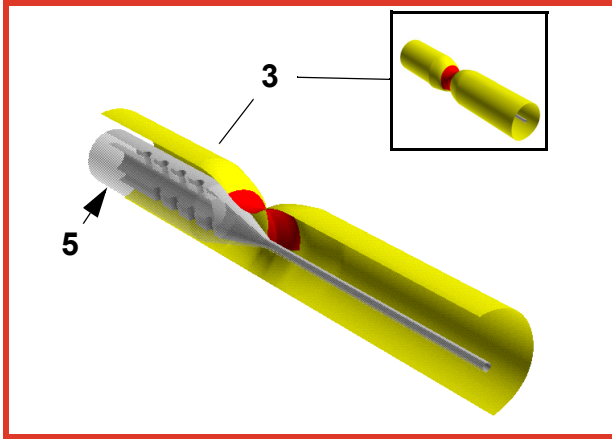
- [How to append two datasets](#)
- [How to display shear on the artery wall](#)
- [How to visualize and analyze data on two datasets in one session](#)
- [How to apply characteristics of one surface to all surfaces of the same type using Unify](#)
- [How to visualize the stress and concentration contours near the catheter holes](#)
- [How to calculate the mass flow rate](#)
- [How to visualize the flow of the medicine](#)

Step 1: Append Two Datasets

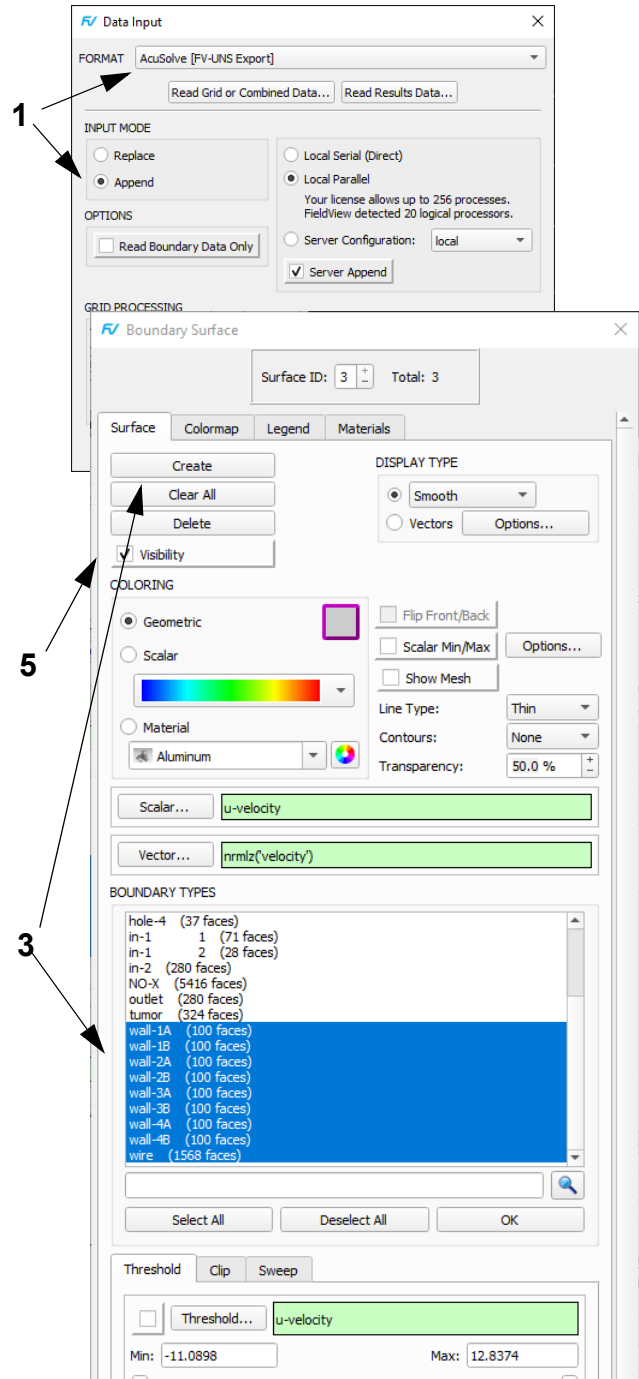
This tutorial looks at application from the biomedical industry. A catheter is inserted into an artery with a tumor. The injection of fluid through the catheter on the artery and tumor is investigated.



The pictures on each page correspond to the final image that should be created if the steps are completed correctly.

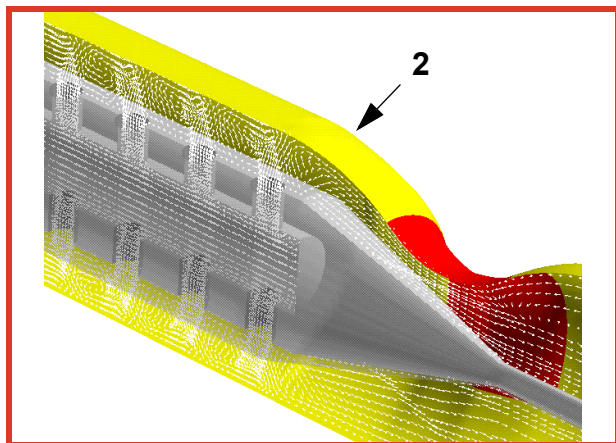


1. Bring up the **File** → **Data Input** → **AcuSolve [FV-UNS Export]** panel. In the default **Replace** mode, read (Grid or Combined Data) **catheter.uns** for one side of the catheter/artery geometry. Switch the **INPUT MODE** to **Append**, read **catheter.uns** again and **Close** the panel. [Page 36 Reference Manual]
2. On the main toolbar, click **Dataset...** Set **SCALE X:** to **-1**, click **Apply** and **Close**. Dataset 2 now mirrors Dataset 1.
3. On the main toolbar, make Dataset 1 current. Bring up the **Boundary Surface** panel, click **Create**, select **BOUNDARY TYPE** **tumor** and click **OK**. Change **DISPLAY TYPE** to **Smooth** shading and **Geometric** **COLORING** to red. **Create** a second surface consisting of **artery-wall 1** and **2** with **Geometric** **COLORING** yellow. **Create** a third surface consisting of **catheter-I** and **catheter-O**, the 8 **wall-** boundaries, and boundary type **wire**, with **Geometric** **COLORING** gray.
4. Bring up **File** → **Save Restart** → **Current Dataset...** and **Save** a Current Dataset restart. Make Dataset 2 current. Bring up **File** → **Open Restart** → **Current Dataset...** and **Open** the Current Dataset restart to create the same three surfaces on Dataset 2 as Dataset 1.
5. On Dataset 2, turn the **Visibility** of the **tumor** and **artery** off and make the **catheter/wall** boundaries **50%** transparent. On the Transform Controls toolbar, turn the Outline display off.

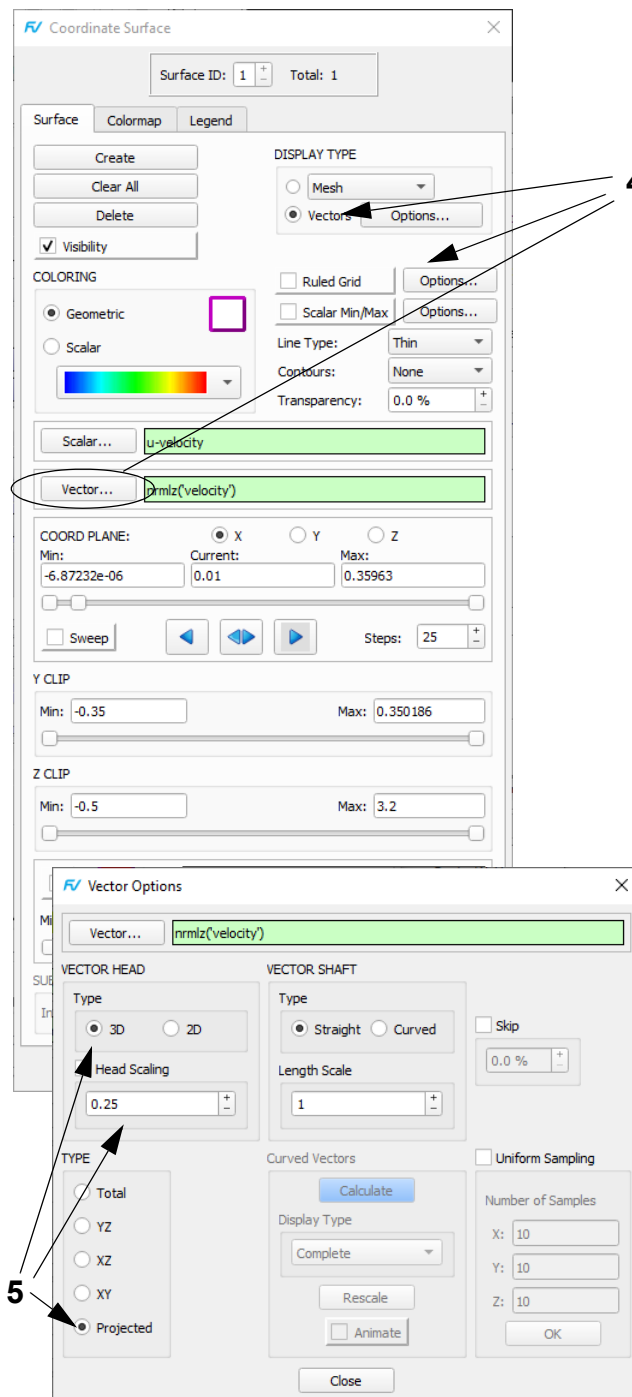


Step 2: Visualize Recirculation Zones

In this step we create a vector coordinate surface to visualize the recirculation zones created by the drug release through the holes of the catheter.

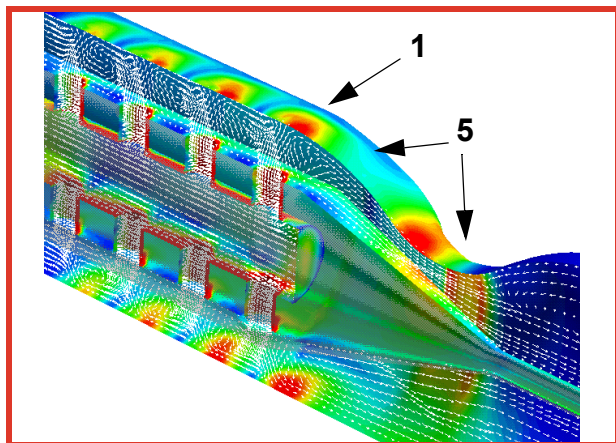


1. Rotate the view slightly and zoom into the **catheter** holes and the **tumor**.
2. “Quick-pick” (double-click) a surface on Dataset 1 (or edit the **Dataset** field on the main toolbar or **Dataset Controls** panel) to make Dataset 1 current.
3. Bring up the **File** → **Open Restart** → **Formula...** browser. In the **biomedical** directory, select **bio.frm** and click **Open**. [\[Page 260 Reference Manual\]](#)
4. **Create an X=0.01 Coordinate Surface**. Change the **DISPLAY TYPE** to **Vectors**. For **Vector Function**, click on **Vector..** and use the **Function Selection** panel to select **nrmlz('velocity')** and click **Calculate**.
5. Click **Options...** to bring up the **Vector Options** panel. Turn on **Head Scaling** and change the value to **0.25**. Change **VECTOR HEAD Type** to **3D** and **TYPE** from **Total** to **Projected**. Close the **Vector Options** panel.
6. Change the **Geometric COLORING** to white. On the side toolbar, click the **Zoom** icon. Use the left mouse button (**M1**) to drag a rectangular zoom box in the vicinity of the catheter holes). Notice the recirculation zones near the holes of the catheter where the drug is released. On the side toolbar, click **Undo Zoom** to reset the view.

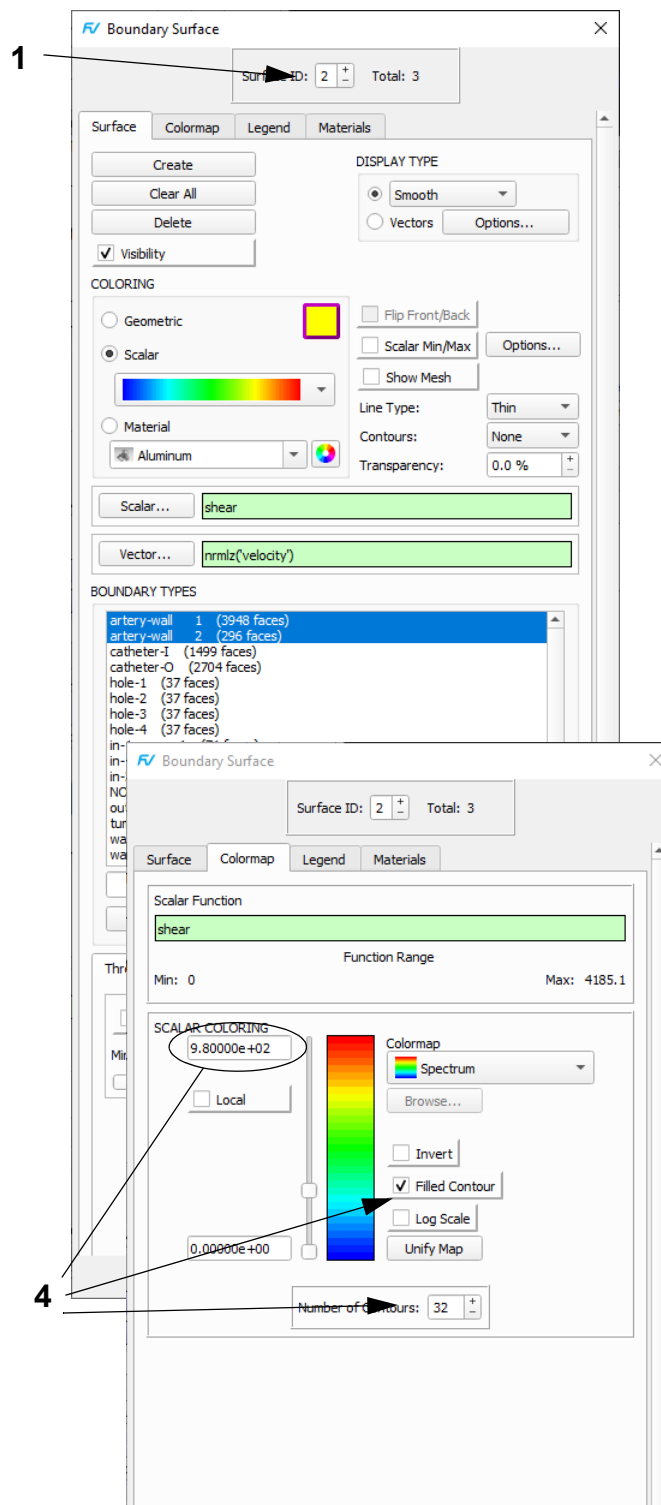


Step 3: Display the Shear on the Artery Wall

In this step we show the shear on the artery wall created by the drug release through the holes of the catheter.

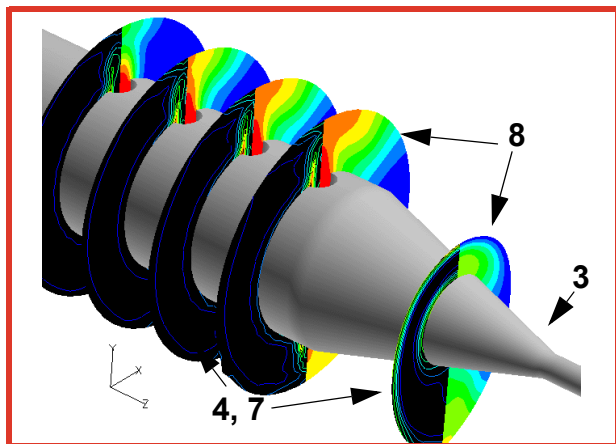


1. "Quick-pick" (double-click) on the **artery wall** OR set the **Dataset** to 1 and, on the **Boundary Surface** panel, the **Surface ID** to 2.
2. For **Scalar Function**, click **Scalar..** then on the **Function Selection** panel, scroll down to select **shear** and click **Calculate**.
3. Change **COLORING** from **Geometric** to **Scalar**.
4. On the **Colormap** tab, change the maximum to **980** and the **Number of Contours** to **32**. Turn on **Filled Contour**.
5. On the Main FieldView menu, select **Tools** → **Unify** to make all the surfaces of the same type (boundary) and of the current dataset (only) display **shear** with the set color ranges. Notice the very high shear rates on the artery wall due to the delivery of the drug through the holes of the catheter. This shows an undesirable amount of shear on the artery. [\[Page 345 Working with FieldView\]](#)

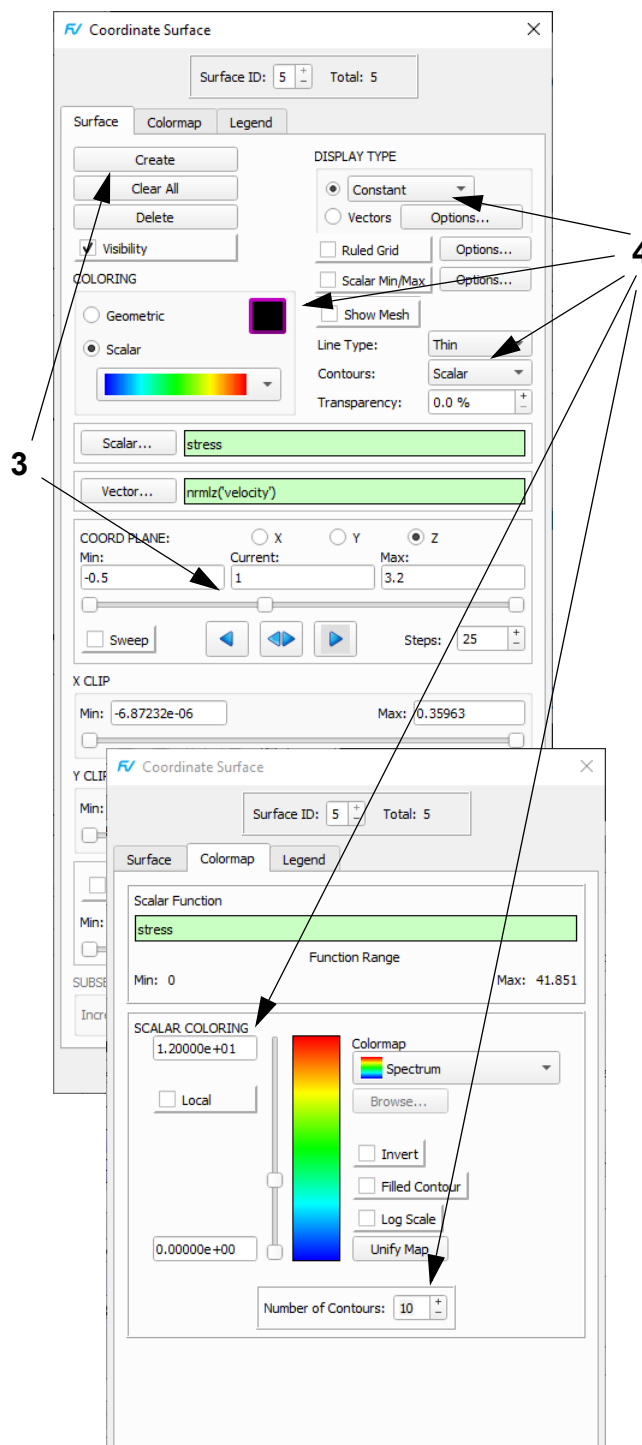


Step 4: Visualize Stress and Concentration Contours

In this step we show stress contours and concentration contours at and near the location of the holes of the catheter. For each set of planes, we show a different way to create multiple surfaces of the same type.

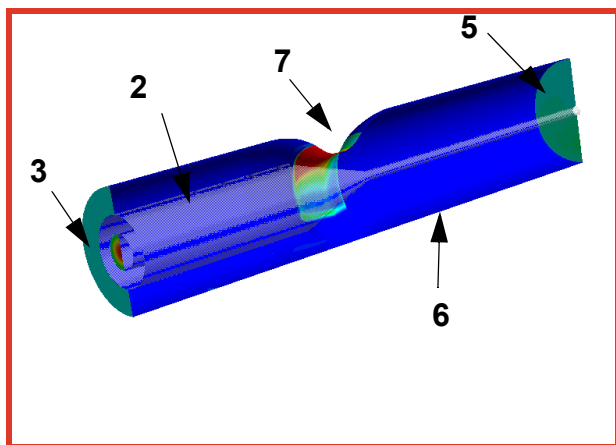


1. Successively “quick-pick” (double-click) and turn the **Visibility** off for the white vector coordinate surface and the **artery wall** and **tumor** boundary surfaces. “Quick-pick” on the **catheter** boundary surface and change to **Geometric COLORING**. Adjust the view similar to that shown above.
2. “Quick-pick” on the **catheter** boundary surface for Dataset 2 and set the **Transparency** back to 0.
3. While Dataset 2 is current, **Create** five coordinate surfaces: at **Z=0.0, 0.2, 0.4, 0.6, and 1.0**. [\[Page 232 Working with FieldView\]](#)
4. Change the **DISPLAY TYPE** of the current surface (**Z=1.0**) to **Constant** shading. Change to **Geometric COLORING**, color **black**. **Contours**: from **None** to **Scalar** and **Scalar Function** to **stress**. On the **Colormap** tab, change the maximum to 12 and the **Number of Contours** to 10.
5. Select **Tools** → **Unify** to make all the surfaces of the same type (coordinate) and of the current dataset (only) display **stress** with the set color ranges.
6. Use **File** → **Save Restart** → **Current Dataset...** to **Save** a Current Dataset restart. Make Dataset 1 current. Use **File** → **Open Restart** → **Current Dataset...** to **Open** the Current Dataset restart to create the same five surfaces on Dataset 1 as Dataset 2.
7. “Quick-pick” one coordinate surface in Dataset 1. Change the **Scalar Function** to **species conc 1** and **COLORING** to **Scalar**. Set **Filled Contours** on the Colormap Tab. Select **Tools** → **Unify** to propagate the change to the other four surfaces. (This step not shown.)

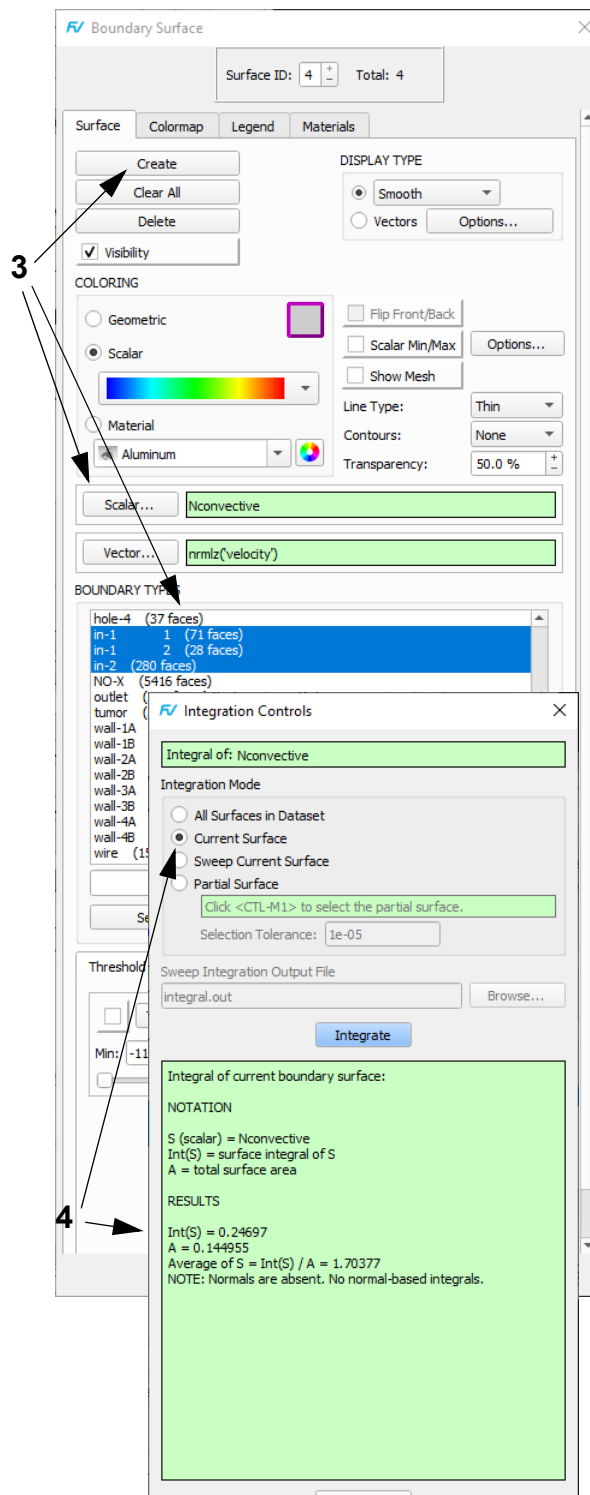


Step 5: Calculate the Mass Balance

In this step we calculate the mass balance of the solution by taking into consideration both the convective flux through the artery as well as the diffusive flux through the artery wall and the tumor.

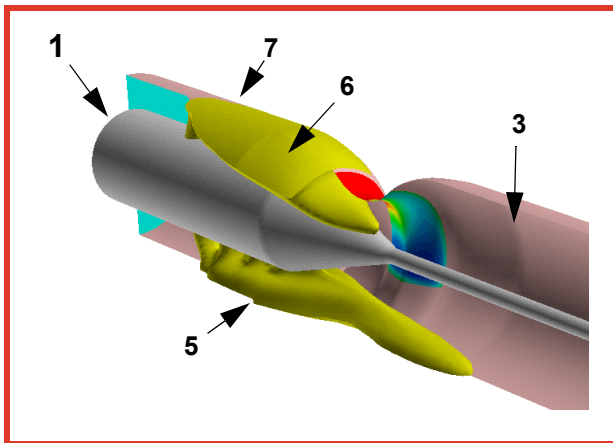


1. **Clear All** coordinate surfaces for both datasets and adjust the view similar to that shown above.
2. For Boundary Surface 3 of both datasets, turn on the **Visibility** and change the **Transparency** to **0.5**. For Dataset 1, Boundary Surfaces 1 and 2, turn on the **Visibility**.
3. For Dataset 1, **Create** a 4th Boundary Surface. Select **in-1 1**, **in-1 2**, and **in-2** and click **OK**. Change **COLORING** to **Scalar**. Change the **Scalar Function** to **Nconvective** to show the convective flux into the artery.
4. Select **Tools** → **Integration...** to bring up the **Integration Controls** panel. Change the **Integration Mode** to **Current Surface**. Click **Integrate** to get a value of **0.24697**.
5. **Create** a 5th Boundary Surface using **outlet**. This surface has the **Scalar Function** already set to **Nconvective**. Click **Integrate**. The convective flux out of the artery is **0.184225**. There is a net loss of mass of $0.24697 - 0.184225 = 0.062745$ inside the artery.
6. For Boundary Surface 2, change the **Scalar Function** (artery wall-1, 2) to **Ndiff-Normal**, the diffusive flux into the wall. On the **Colormap** tab, change the min and max to **0** and **0.5**. **Integrate** to get **~0.0065664**.
7. Switch to Boundary Surface 1. Change the **Scalar Function** to **Ndiff-Normal**. Turn on **Visibility**. **Integrate** to get **~0.055759**. This gives a total of **~0.062325**. Thus **mass balance** is less than 1%. We see only a small amount of drug actually gets to the **tumor**. Most of it is washed away down the artery.



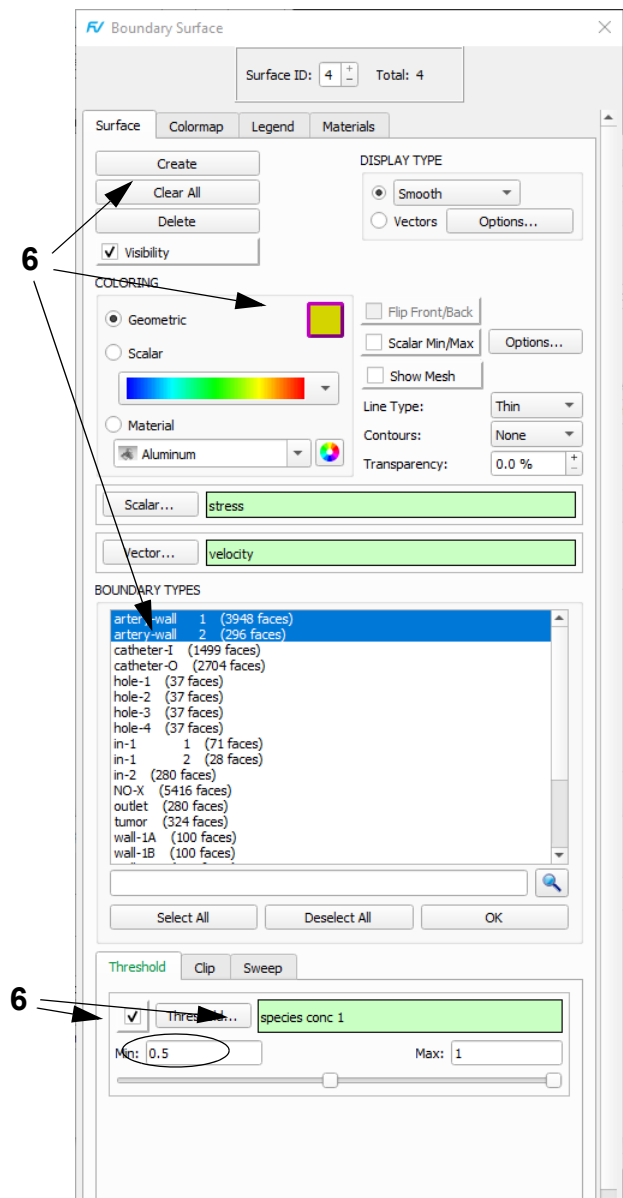
Step 6: Visualize the Drug Delivery

In this step we look at the flow of the medicine and show some visualization “tricks”.



1. Turn visibility ON for Dataset 2, Boundary Surface 3 (the **catheter**), set the **Transparency** to 0.
2. Adjust the view similar to that shown above.
3. “Quick-pick” the **artery wall** (Dataset 1, Boundary Surface 2) and change the **COLORING** to **Geometric** (yellow).
4. Bring up **Tools** → **Color Mixer**. Click on the yellow chip. Change the **Red**, **Green** and **Blue** values to **235**, **182**, and **180**, respectively. Click **Apply** and **Close**.
5. Change to Dataset 2 and **Create** an **Iso-Surface** of **species conc 1**. Set the **Iso Function Current**: value to **0.5** and make the color blue. Change the **DISPLAY TYPE** to **Smooth** shading. Bring up the **Color Mixer** and change the blue color chip **RGB** values to **212**, **212**, **0**. Click **Apply** and **Close**.
6. The iso-surface intersecting the **artery wall** is “open”. To “close” it, **Create** a 4th Boundary Surface on Dataset 2 consisting of **artery-wall 1** and **2**. Color it “dark yellow” used in step 5. Click **Threshold...** then on the **Function Selection** panel, select **species conc 1** and click **Calculate**. Turn on the Threshold checkbox to activate, and set **Min**: to **0.5** to fill in the “open top” and clip the rest.
7. Make sure Dataset 2 is current. On the **Dataset Controls** panel, set **DUPLICATION** to **Mirror**, selecting only **X** of the three axis (Y, Y, and Z) shown on the panel and **Close** the panel.
8. Due to the existence of coincident surfaces, we sometimes see a problem where two surfaces “fight” for dominance. To resolve this, “quick-pick” on the flesh-colored artery wall in dataset 1. We know that the

newly-created iso-surface is shown where **species conc 1** is between 0.5 and 1.0. **Threshold** the flesh-colored boundary surface with **species conc 1**, but this time threshold it so that only the portion between 0.0 and 0.5 shows, thus eliminating the portion in the same location as the iso-surface.



Chapter 11

Chemical Processing Tutorial

11

What You Will Learn

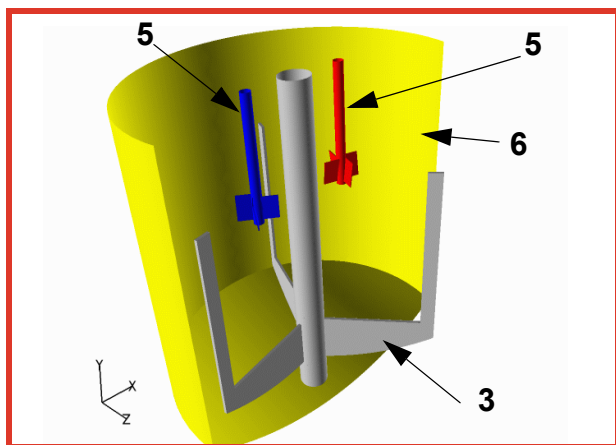
- [How to analyze flow around the shafts](#)
- [How to automatically add streamline seeds between two seed points](#)
- [How to adjust presentation rendering](#)
- [How to increase integration accuracy](#)
- [How to calculate the impeller flow number](#)

Step 1: Read in FV-UNS Data

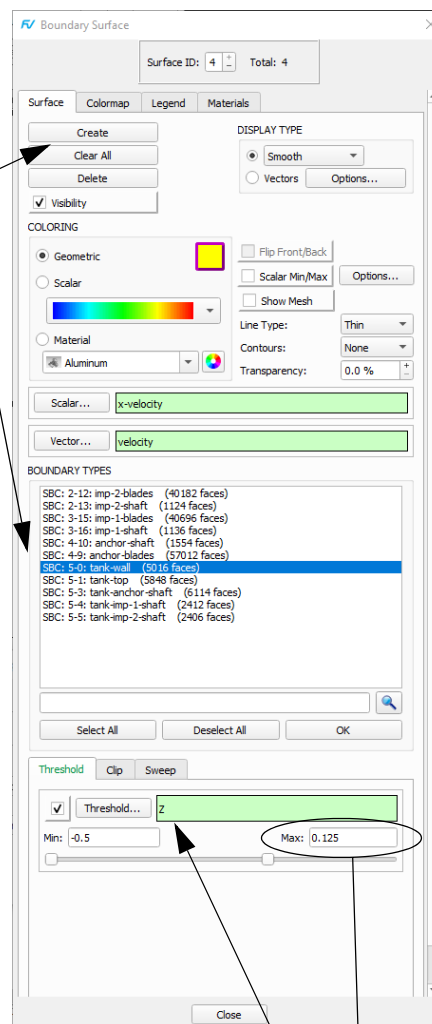
This tutorial examines a three blade mixer. In this step, the geometry of the problem is visualized.



The pictures on each page correspond to the final image that should be created when the steps are completed correctly.

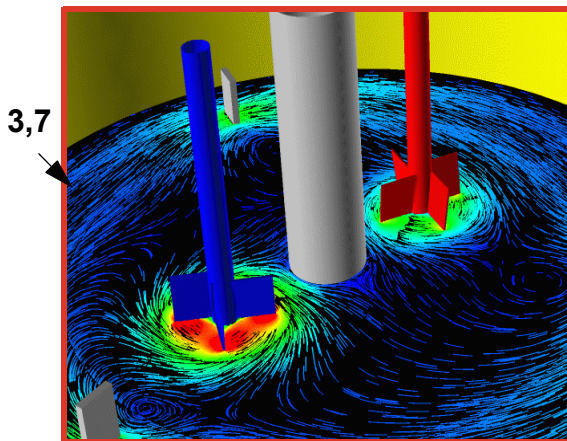


1. Bring up **File** → **Data Input** → **FLUENT [FV-UNS Export]**. Click **Read Grid or Combined Data...** and open **anchor_mixer.fv**. Close the **FLUENT [FV-UNS Export]** panel. [Page 36 Reference Manual]
2. On the **Transform Controls** toolbar, click the icon to toggle **Outline** off.
3. Click the **Bound** icon on the side toolbar to bring up the **Boundary Surface** panel and click **Create**. Hold down the **Ctrl** key to select **SBC: 4-9: anchor-blades**, **SBC: 4-10: anchor-shaft** and **SBC: 5-3: tank-anchor-shaft**. Click **OK**. Change the **DISPLAY TYPE** to **Smooth** shading. [Page 240 Working with FieldView]
4. On the **Transform Controls** toolbar, change **Action:** from **Multi-transform** to **Locked Rotation** and use left and right mouse to orient the model. Change **Action:** back to **Multi-Transform** and use right mouse to zoom into the model.
5. **Create** a second boundary surface with **SBC: 2-12: imp-2-blades**, **SBC: 2-13: imp-2-shaft** and **SBC: 5-5: tank-imp-2-shaft**. Change the **Geometric COLORING** to blue. **Create** a third surface with **SBC: 3-15: imp-1-blades**, **SBC: 3-16: imp-1-shaft** and **SBC: 5-4: tank-imp-1-shaft**. Change **Geometric** to red.
6. **Create** a fourth surface with **SBC: 5-0: tank-wall**. Change **Geometric** to yellow. For **Threshold Function**, click **Select...** On the **Function Selection** panel, select **Z** and click **Calculate**. Change **Max:** to **0.125** and turn **Threshold Clip** on.

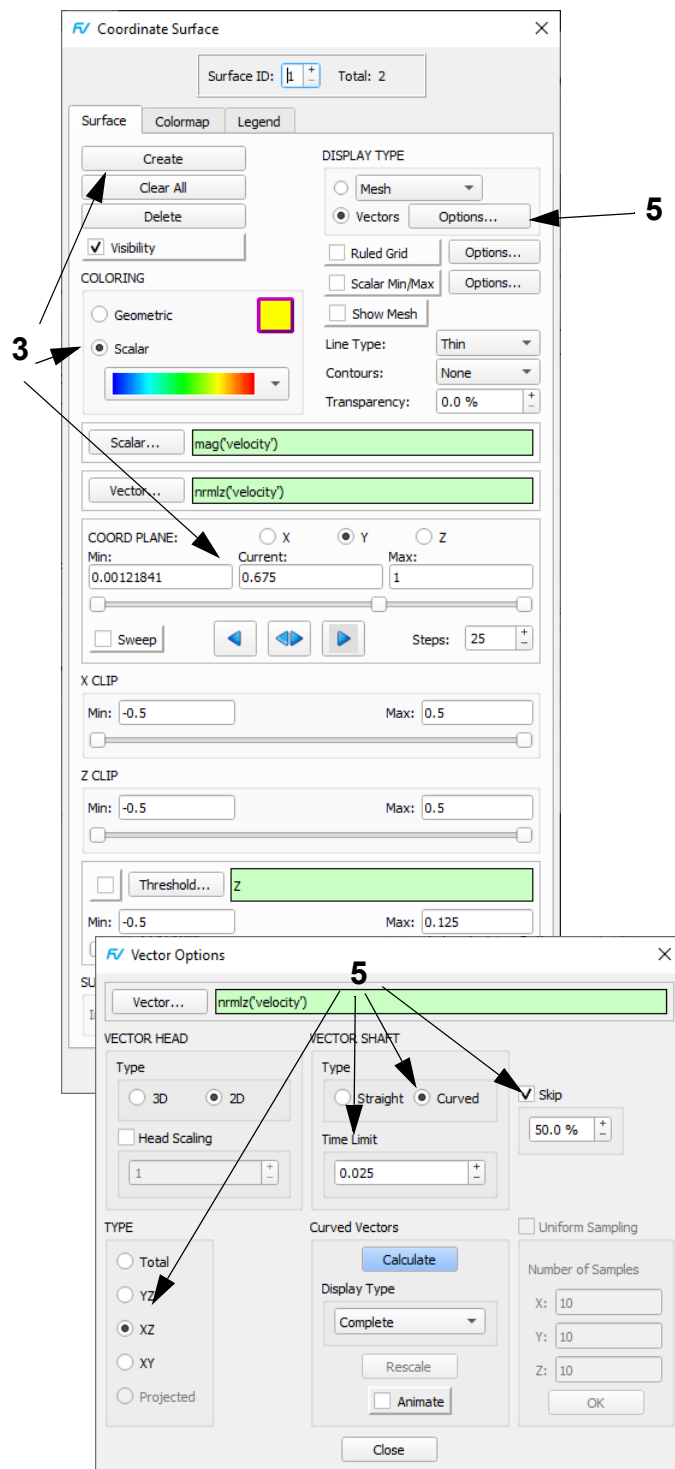


Step 2: Analyze Flow around the Rotating Shafts

In this step we examine the flow in the plane of the smaller blades. We create a vector coordinate surface to show the flow directions. A coincident black coordinate surface highlights the vectors. Notice how the flow is clockwise around the red blade and counterclockwise around the other two.

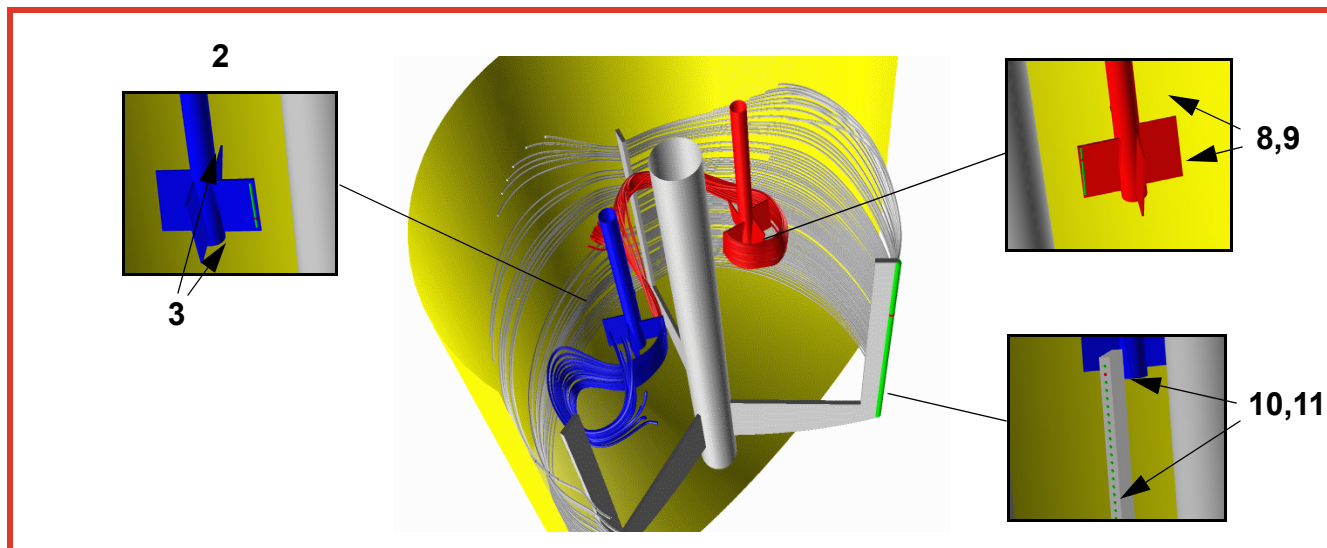


1. Click the side toolbar **Zoom** icon and drag a box to zoom in to the smaller blades. Use the middle mouse button to rotate the view down. Open the **Viewer Options** panel and turn **Axis Markers** off. Close the **Viewer Options** panel.
2. Bring up the **File** → **Open Restart** → **Formula...** browser and open **updated_chem.frm**. [\[Page 260 Reference Manual\]](#)
3. On the side toolbar, click the **Coord** icon to bring up the **Coordinate Surface** panel. **Create** a **Y** surface at **0.675**. Change the **DISPLAY TYPE** to **Vectors**. For **Vector Function**, click **Select...** On the **Function Selection** panel, select **nrmlz('velocity')** and click **Calculate**. Change **COLORING** to **Scalar**. For **Scalar Function**, **Select...** **mag('velocity')** and **Calculate**.
4. On the **Colormap** tab, change the maximum to **0.8**.
5. Click **Vectors Options...** Turn **Skip** on. Increase the **Skip Value** to **50%**. Change the **VECTOR SHAFT** from **Straight** to **Curved**. Change the **Time Limit** to **0.025** and **Calculate**. Change the **TYPE** from **Total** to **XZ**. Close the **Vector Options** panel.
6. On the **Surface** tab, **Create** another coordinate surface. Change the **DISPLAY TYPE** to **Constant** shading and the **COLORING** to **Geometric**. Change the **Geometric** color to black.

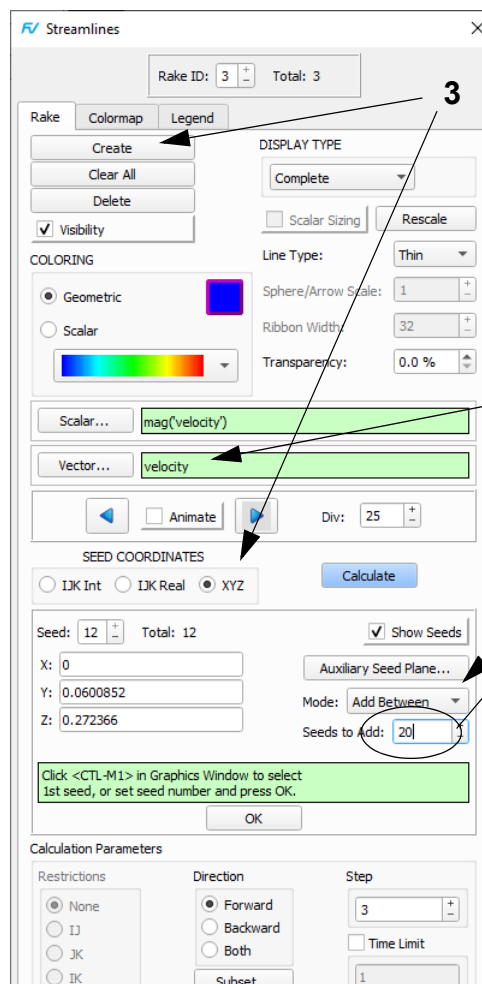


Step 3: Examining Mixing Patterns

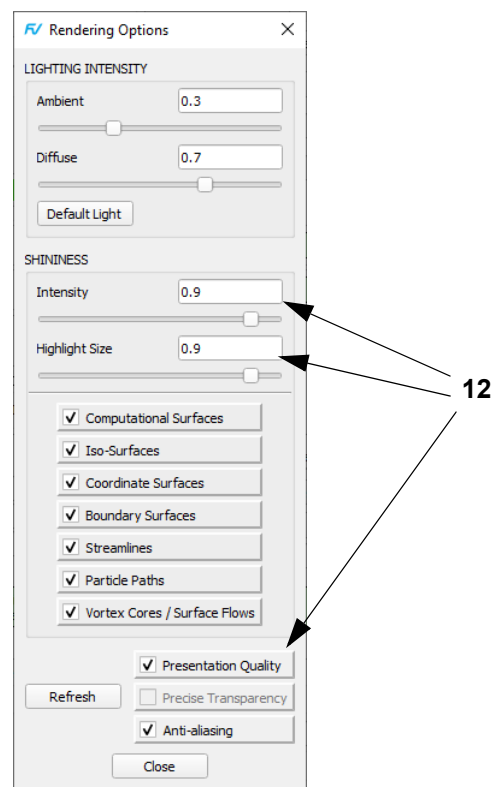
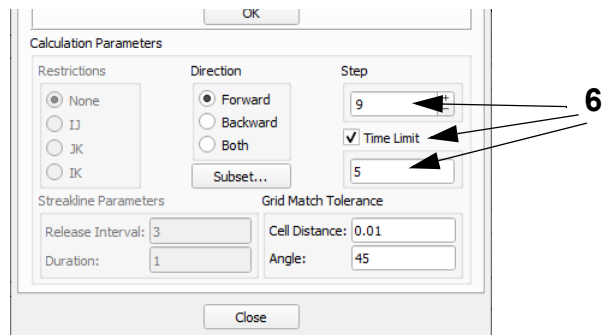
In this step we examine the mixing patterns and look at residence time using time restricted streamlines.



1. Zoom back out and rotate the model back up. “Quick-pick” (double-click) on both coordinate surfaces to turn **Visibility** off. [\[Page 75 User's Guide\]](#)
2. Click the side toolbar **Zoom** icon. Drag a box to zoom in to the blue blade/shaft to make seeding easier.
3. Bring up the **Streamlines** panel, click **Create** and note **SEED COORDINATES** are **XYZ**. In default **Mode: Add**, <Ctrl + left click> near the top of the blade element. Add a second seed near the bottom of the blade element. [\[Page 184 Working with FieldView\]](#)
4. Change **Mode:** from **Add** to **Add Between**. Note **Seed: 2** (at the bottom) is current and click **OK**. Click the minus sign to change the current **Seed:** to **1** and click **OK**. Two end points for between seeding are now defined. Change **Seeds to Add:** from **10** to **20** and click **OK**. There should now be 22 seeds (21 green and the current seed red) in a vertical line on the blue blade element; if not, **Delete** the current rake and go back to 3 to try again until you succeed with this step.
5. You will need to change the **Vector Function**. For **Vector Function**, click the **Vector..** button, then on the **Function Selection** panel, select **velocity** and click **Calculate**.

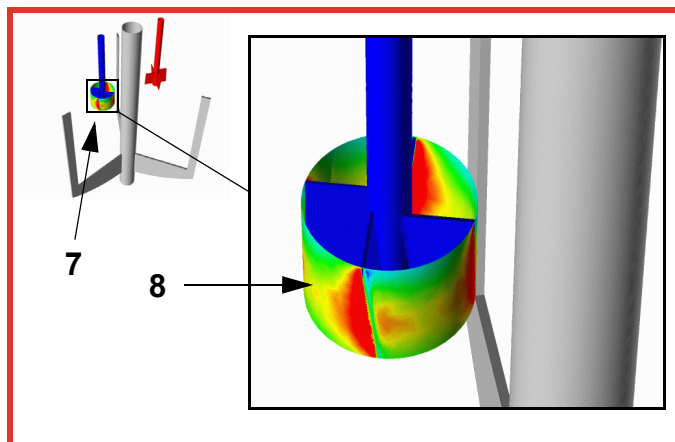


6. In the **Calculation Parameters** section, change the **Step** value to **9** to increase the integration accuracy. Turn **Time Limit** on and edit its value to **5** to limit the streamlines duration to **5** time units. Click **Calculate**.
7. Change the **Geometric COLORING** to blue to match the blade element.
8. Set up the view for the next rake creation. With **Action:** set to **Multi-Transform** on the **Viewer Toolbar**, use right mouse to zoom back out to see the second blade. Click the side toolbar **Zoom** icon and drag a box to approximate the inset view.
9. **Create** a second rake, this time on impeller 1. Use the top and bottom seeding technique with **Add Between** (20 seeds) described in 4. Change the **Geometric** color to red to match the shaft/blade. Click **Calculate**.
10. Zoom back out and in to approximate the inset view.
11. **Create** a third rake on the anchor/shaft boundary surface. Use the same top, bottom and **Add Between** seeding technique with **80** seeds. Change the **Geometric** color to gray to match the anchor/shaft. Click **Calculate**. Transform back to the beginning view.
12. Bring up the **View → Rendering Options...** panel. Adjust the **SHININESS Intensity** and **Highlight Size** to about **0.9** and turn **Presentation Quality** on. The streamlines should become “shiny” stream tubes and the surfaces should also become “shiny.” On the transform toolbar, change the **Object:** from **World** to **Light** and transform the light direction to obtain a suitable view. **Close** the **Rendering Options** panel. [\[Page 58 Working with FieldView\]](#)

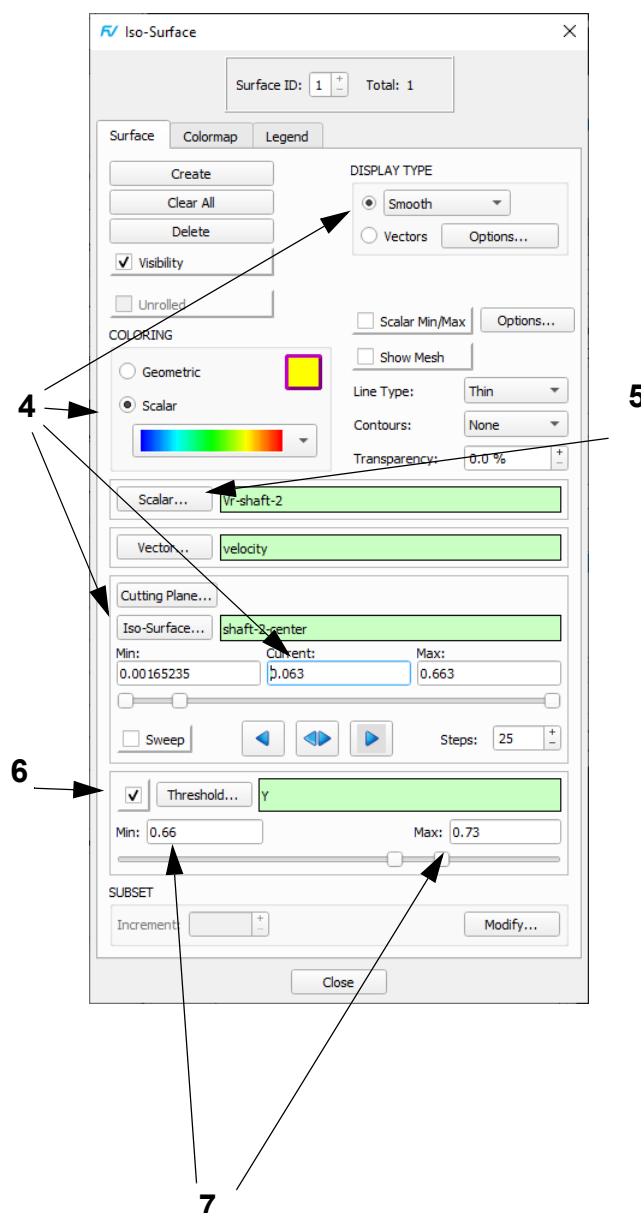


Step 4: Use the Function Calculator to Compute the Flow Number

This step uses **FieldView's** powerful function calculator and integrator to compute the *Flow number (F)* for the two impellers. The functions used in this step were read in previously as part of the Formula Restart file and are defined at the end of this tutorial. [See “Equations for Chemical Processing” on page 164.]



1. “Quick-pick” any of the streamline rakes to make them current. Either turn **Visibility** and **Show Seeds** off for all 3 streamlines one at a time OR press **Clear All** to delete all of the rakes at once. [Page 75 User's Guide]
2. With **Action:** set to **Multi-Transform** on the **Viewer Toolbar**, use the middle mouse to rotate the model down slightly.
3. “Quick-pick” on the yellow **tank-wall**. On the **Boundary Surface** panel, turn **Visibility** off.
4. Bring up the **Iso-Surface** panel. Click **Create** and **Define Iso-Function...** On the **Function Selection** panel, select **shaft-2-center** and click **Calculate**. Change **COLORING** to **Scalar** and **DISPLAY TYPE** to **Smooth** shading. [Page 170 Working with FieldView]. Change the **IsoFunction Current:** value to **0.063**.
5. For **Scalar Function**, click **Scalar...** then on the **Function Selection** panel, select **Vr-shaft-2** and click **Calculate**. Similarly, On the **Colormap** Tab, change the minimum and maximum to **-0.13** and **0.25**, respectively.
6. Change the **Threshold Function** to **Y** and activate by clicking the checkbox **ON**. [Page 100 User's Guide]
7. Set threshold range **Min:** to **0.66** (this is **h1** - see following section) and **Max:** to **0.73** (this is **h2**).



8. Zoom in to the blade element as shown above. Note that the iso-surface has been thresholded to span only between the bottom and the top of the blades.
9. Navigate to **Tools** → **Integration...** to bring up the **Integration Controls** panel. Change the **Integration Mode** to **Current Surface** and click **Integrate**. The result for the pumping capacity, **Q**, is the value of **integral**, which is $\sim 0.00377 \text{ m}^3 \cdot \text{s}^{-1}$.
10. Perform the following hand-calculation for blade2:
[See “Equations for Chemical Processing” on page 164.]

$$FI = Q / (N \cdot D^3)$$

with

$$N = 20 \text{ rads/sec (} = 3.183 \text{ sec}^{-1}\text{)}$$

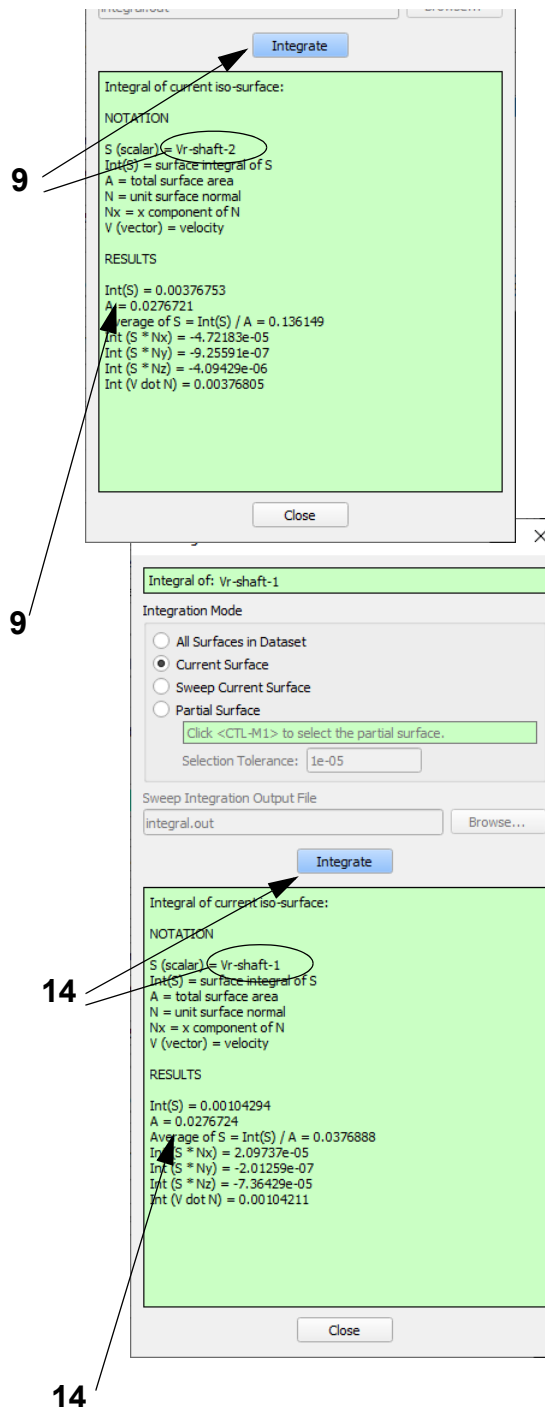
$$D = (0.063 \times 2), \text{ blade diameter}$$

so

$$FI = 0.592$$

A Rushton impeller has a Flow number of around **0.75**. The impeller in this problem has angled blades which reduce the radial flow speed, so **0.592** is reasonable.

11. Zoom back out, then zoom into the right (red) blade.
12. The following use the iso-surface attributes set in steps 4-10 to repeat the calculation on the red shaft. To move the iso-surface to the red shaft, change the **Iso Function** to **shaft-1-center**, the **Scalar Function** to **Vr-shaft-1** and the **Current:** value to **0.063** (these and the following steps are not represented on the panels to the right).
13. On the **Colormap Tab**, change the minimum and maximum to **-0.3** and **0.3**, respectively.
14. Integrate the second blade as in step 9 to get a pumping capacity, **Q**, integral value of $\sim 0.00104 \text{ m}^3 \cdot \text{s}^{-1}$, which is one third of the pumping capacity of the identical blue blade operating at double the rotational speed.
15. The red blade has a rotational speed **N** of **10 rads/sec**. We get a Flow number of **0.335**. The pumping capacity (**Q**) of the red blade is significantly less that that of the blue blade.



Equations for Chemical Processing

A number of mathematical expressions were used to create the visualizations and perform the calculations presented in this tutorial and are provided in a **Formula Restart (updated_chem.frm)**. The formulation for calculating the *Flow number*, *Fl*, is given below.

The *Flow number* is given by:

$$Fl = \frac{Q}{ND^3}$$

where the impeller pumping capacity, **Q**, is given by

$$\begin{aligned} Q &= \int_{-h^*}^{h^*} 2\pi \cdot \bar{V} \cdot dz \\ &= \int_{-h^*}^{h^*} 2\pi \cdot V_r \cdot dz \\ &= \int_0^{2\pi} \int_{-h^*}^{h^*} V_r \cdot r dr \cdot dz \\ &= \iint_A V_r \cdot dA \end{aligned}$$

V_r is the radial mean velocity component and h^* and $-h^*$ are the points above (h_2) and below (h_1) the disk where the radial mean velocity equals zero. Since **FieldView** computes area integrals when integration is performed, we have transformed the initial definition of **Q** into an area integral. It is this quantity that **FieldView** is integrating in the tutorial.

The quantities used for the calculation for the two different blades/shafts are:

	Blade/Shaft1	Blade/Shaft2
h1	0.66	0.66
h2	0.73	0.73
shaft-center	-0.163	0.163
blade radius	0.063	0.063
shaft radius	0.0163	0.0163
rotation rate (Ω rads/s)	20	10

Chapter 12

12

Automotive

Diesel

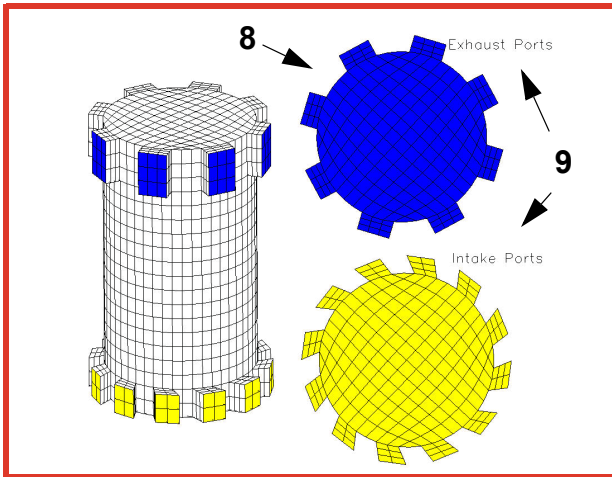
Combustion Tutorial

This tutorial examines the opposed dual-piston problem. The first step visualizes the model geometry. We also show how to make slices through the model and move them. See the KIVA Instructions at the end of this chapter for a description of modifying and converting KIVA data for use in **FieldView**.

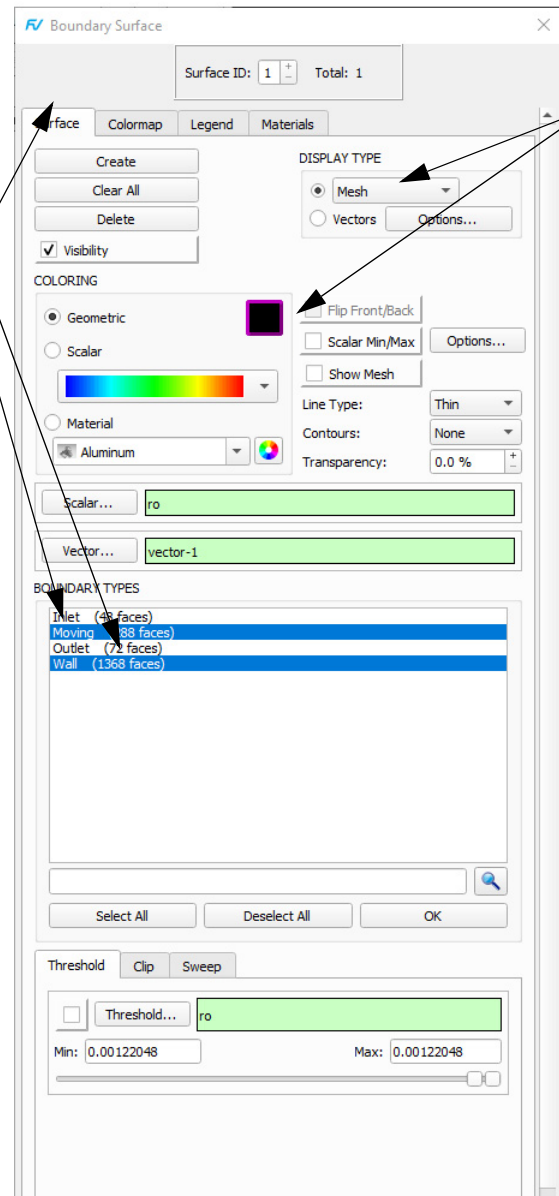
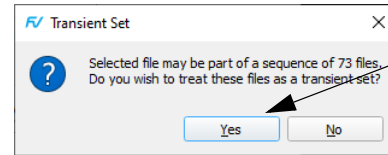
What You Will Learn

- [How to create a hidden line view of the Boundary Surface](#)
- [Display velocity vectors and the gasoline concentration as a function of time](#)
- [How to amplify the contour range of the gasoline](#)
- [How to make an animation of the transient spray results](#)
- [How to make 2D plots of the gasoline and O₂ concentrations](#)
- [Displaying the exhaust mass flow rate as a function of crank angle](#)

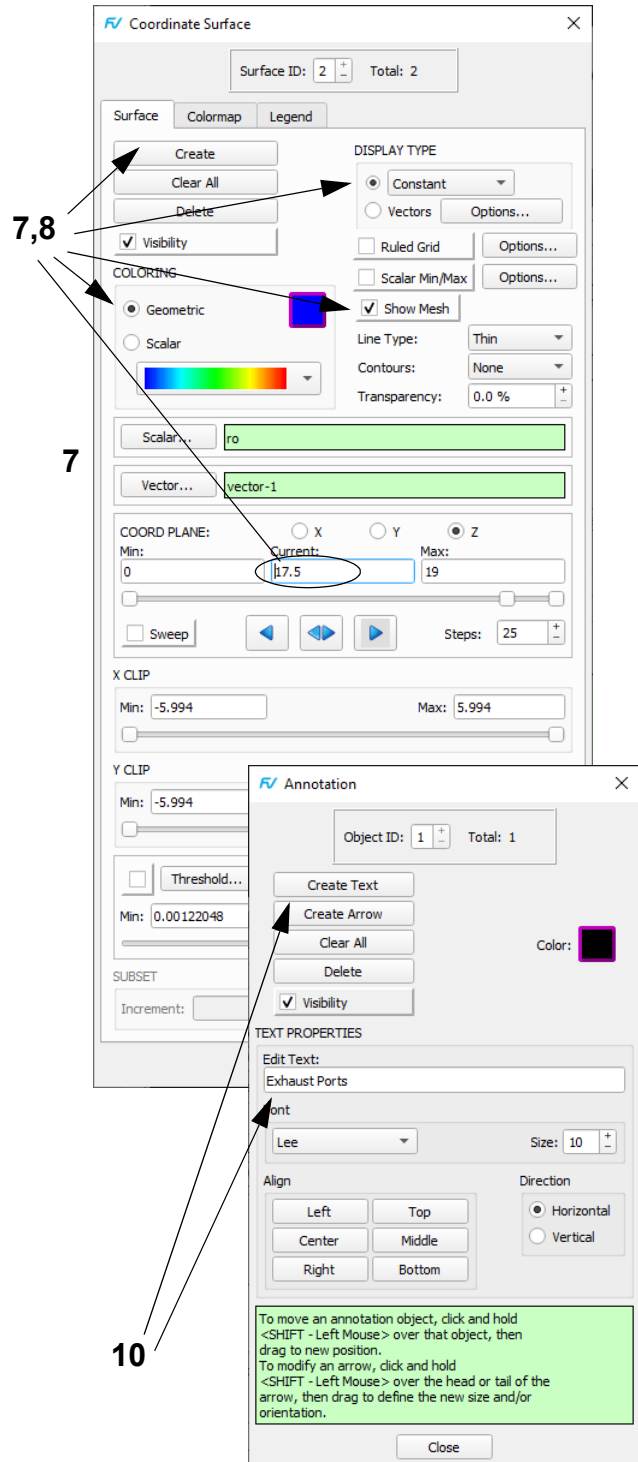
Step 1: Create a Hidden Line View



1. Bring up **File** → **Data Input** → **CONVERGE [FV-UNS Export]**, click **Read Grid or Combined Data...**, select one of the files in the **FieldView** Unstructured transient dataset called **ex4p0001.uns, ..., ex4p0073.uns** and click **Open**. Click **Yes** to treat all 73 files as a transient set. After read-in, a rectangular outline of the model geometry is displayed. **Close** the **CONVERGE [FV-UNS Export]** panel.
2. From the **View** menu, bring up the **Background Color** panel and click the white color chip to change the background color to white.
3. To visualize the geometry, click the side toolbar **Bound** icon or use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. Hold down the Ctrl key to select **Wall** and **Moving**. Click **OK**. Click the **Geometric** color chip and change the color to black. On the **Viewer Options** panel, turn **Outline** and **Perspective** off.
4. On the **Boundary Surface** panel, click **Create** and select **Inlet**. Change the **DISPLAY TYPE** from the default setting of **Mesh** to **Constant** shading. Turn **Show Mesh** on and change the **Geometric COLORING** to yellow. Click **Create** and select **Outlet**. Change the **Geometric COLORING** to blue.
5. To produce a hidden line view of the geometry, **Create** a fourth boundary surface and choose the **Wall** and **Moving** boundaries. Change the **Geometric** color to white.

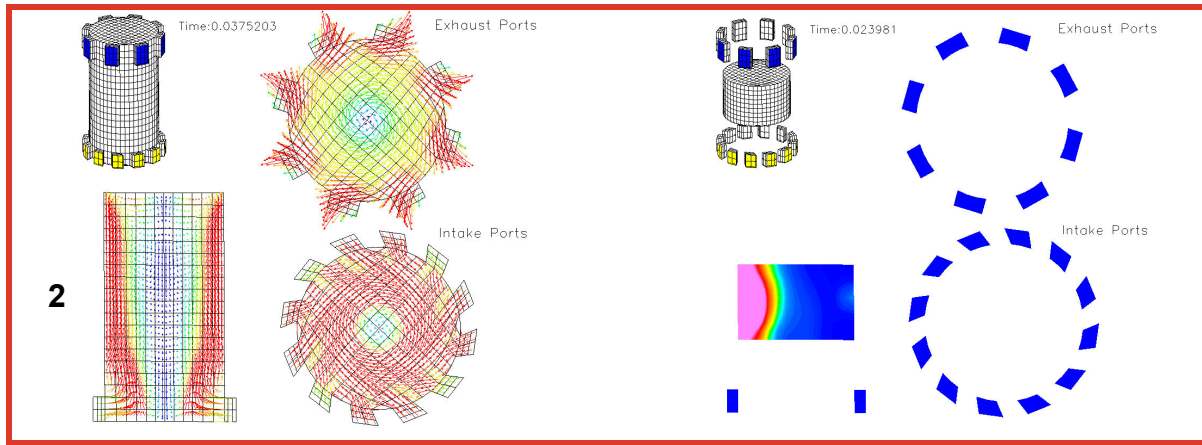


6. Use the **Viewer Toolbar** transform controls to rotate, translate and zoom the dual piston assembly to approximately match the figure.
7. To study the area near the exhaust and intake ports, click the side toolbar **Coord** icon or use the **Visualization Panels** menu to bring up the **Coordinate Surface** panel. Click **Create**. Change the **COORD PLANE** to **Z** and the **Current:** value to **1.0**. Set the **DISPLAY TYPE** to **Constant** shading and turn **Show Mesh** on. Change the **Geometric COLORING** to yellow. The surface will not be visible until transformed in Step 9. [\[Page 232 Working with FieldView\]](#)
8. **Create** a second **COORD PLANE** at **Z = 17.5**. Change the **Geometric COLORING** to blue.
9. On the **Viewer Toolbar**, change the transformation **Object:** from **World** to **Surface**. Use the mouse to zoom and position the surfaces as shown.
10. To add labels, click the side toolbar **Anno** icon or use the **Visualization Panels** menu to bring up the **Annotation** panel. Click **Create Text**. On the **Annotation Create** panel, enter label text *Exhaust Ports* and click **OK**. Click the **Color:** chip, change to black and **Close** the **Geometry Color** panel. As instructed on the **Annotation** panel, use Shift + left mouse to reposition the label. Similarly, **Create Text** for and reposition the second label *Intake Ports*. "Quick-pick" (double-click) to switch between labels if necessary. [\[Page 224 Working with FieldView\]](#)
11. To reuse this orientation of the geometry and cutting planes, bring up **File** → **Save Restart** → **Complete**. For **File name**, type **computing_mesh** and Enter or click **Save**.



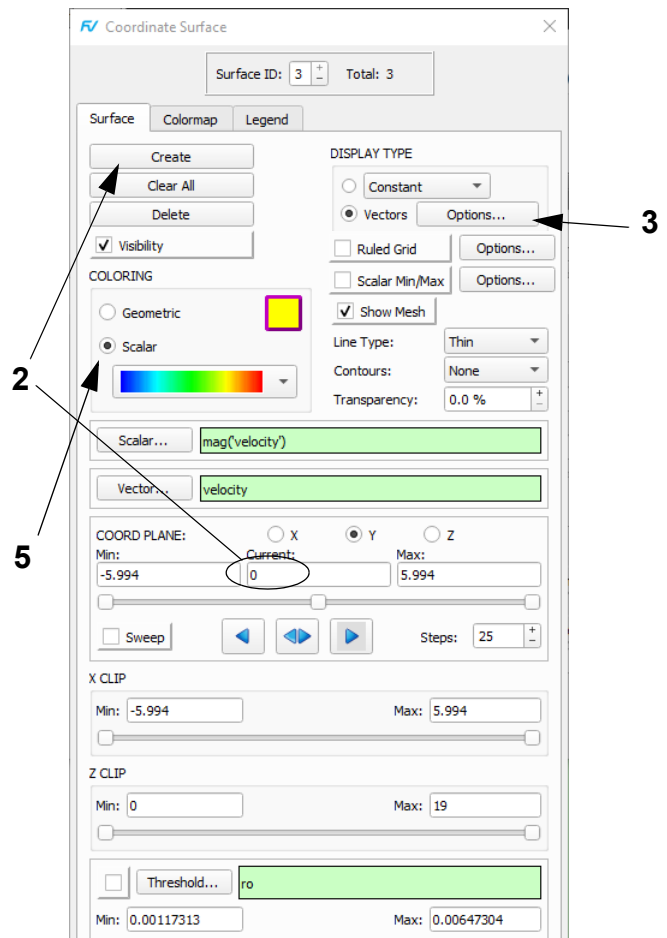
Step 2: Viewing Velocity Vectors and Fuel Concentration

In this step, we show how to view velocity vectors at various points within the model. We also show how to plot gasoline concentration contours and create a simple animation of this transient data.

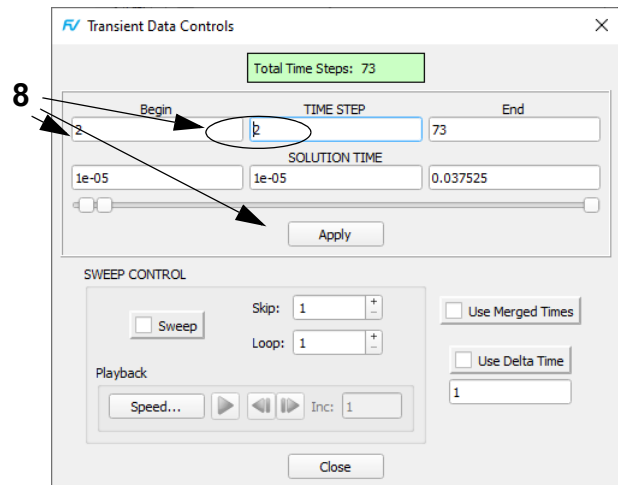
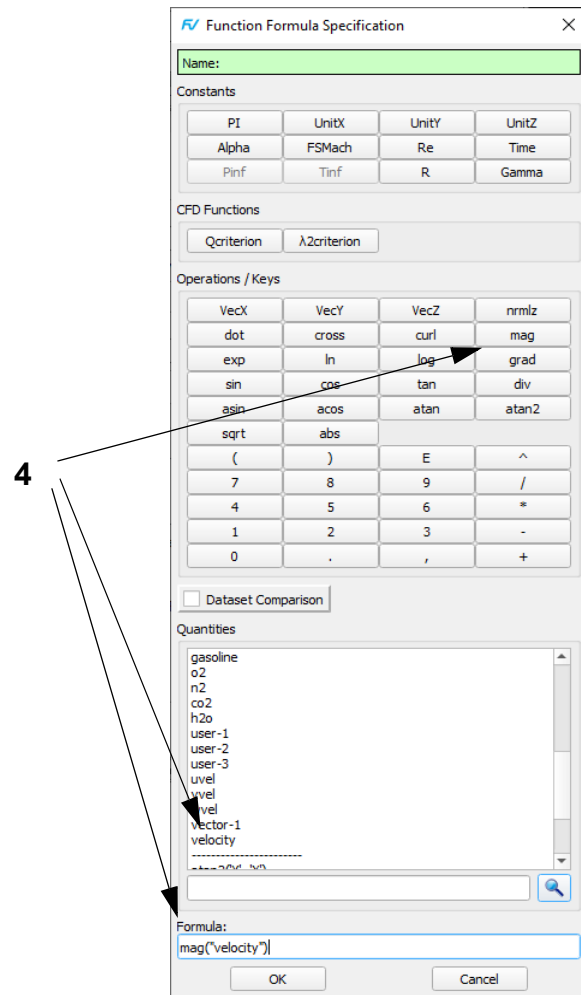


1. With **Object:** set to **World** on the **Viewer Toolbar**, zoom out and reposition the piston model to approximate the upper left view in the figure above. Change **Object:** to **Surface**. “Quick-pick” and reposition each coordinate surface to produce the figure above (not including the coordinate slice at the lower left, which will be created in Step 2).
2. Bring up the **Coordinate Surface** panel and **Create** a third coordinate surface. Set **COORD PLANE** to **Y** with **Current:** value **0.0**. With **Object:** **Surface**, transform and make the new surface visible as shown. NOTE: Depending on the geometry's orientation, surfaces may be rotated compared to above. Click on **Vector..** and select **Velocity** as your vector quantity.
3. Change the **DISPLAY TYPE** to **Vectors** and click **Options...**. On the **Vector Options** panel, increase the **Vector Scale** to **8**. Set the **TYPE** to **Projected**. Turn **Uniform Sampling** on and adjust the **Number of Samples** for **X:**, **Y:** and **Z:** to **40**. Click **OK** and **Close** the **Vector Options** panel.
4. Click the main toolbar **f_x** icon to bring up the **Function Specification** panel. Click **Create...** to bring up the **Function Formula Specification** panel (“CFD Calculator” - see panel next page). In **Operations/Keys**, click **mag**. Scroll down **Quantities** and select **velocity**. In **Operations/Keys**, click **)** to complete the **Formula: mag(“velocity”)**. Click **OK**. On the **Function Name Input** panel, click **Use Formula**. On the **Function Specification** panel, click **Scalar...**. On the **Function Selection** panel, select **mag(“velocity”)**

and click **Calculate**. **Close** the **Function Specification** panel.

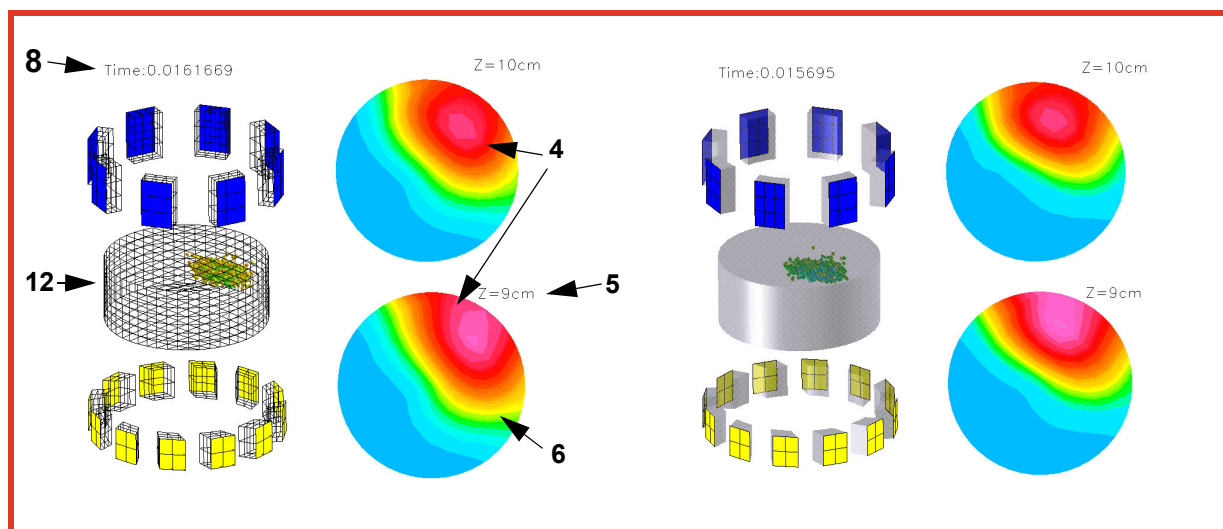


5. Change the **COLORING** to **Scalar**. On the **Colormap** tab, enter a maximum of **5.4e+03** and a minimum of **1.3e+03**. On the main toolbar, click **Unify** to propagate these settings and other attributes of the current coordinate surface to all other coordinate surfaces.
6. Navigate to **Tools** → **Transient Data...** Move the slider or enter the value to set the current **TIME STEP** to **8** and click **Apply**. Close the **Transient Data Controls** panel.
7. Bring up the **Annotation** panel and click **Create Text**. On the **Annotation Create** panel, type **Time: %%T** to create a label showing the Solution Time. Click **OK**. Reposition the label following the instructions on the **Annotation** panel.
8. Reopen the **Transient Data Controls** panel to animate the sequence. Set **Begin** and current **TIME STEP** to **2** and click **Apply**. Choose **Tools** → **Flipbook Build Mode**. Click **OK** to dismiss the **Flipbook Display Size Warning** panel. On the **Transient Data Controls** panel, turn **Build** on. When the building is complete, the **Flipbook Controls** panel may be used to play or **Save** the flipbook.
9. “Quick-pick” one of the coordinate surfaces. On the **Surface** tab, change **DISPLAY TYPE** from **Vectors** to **Constant** and turn **Show Mesh** off. For **Scalar Function**, click **Select...** On the **Function Selection** panel, select **gasoline** and click **Calculate**. On the **Transient Data Controls** panel, change the current **TIME STEP** to **47** and click **Apply**. On the **Colormap** tab, change the **Colormap** to **NASA-1**, the minimum value to **0.0** and the maximum to **5e-05**. On the main toolbar, click **Unify** to apply these values to all coordinate surfaces.
10. Follow the instructions in Step 8 to build a flipbook for **gasoline** concentration.

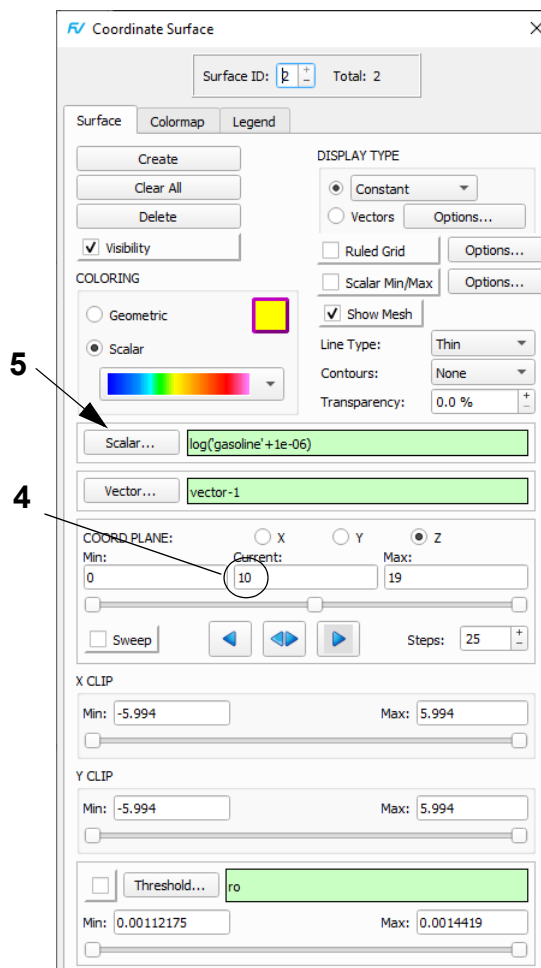


Step 3: Amplifying Contour Range and Animating Spray Results

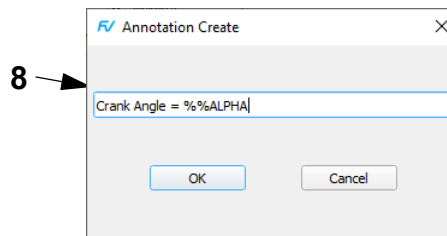
Here we show how to amplify the contour range of the gasoline and read in spray results from KIVA in the form of a Particle Path file. The spray will also be animated.



1. Bring up **File** → **Open Restart** → **Complete**, select **computing_mesh** (the restart saved at the beginning of this tutorial) and click **Open**.
2. On the **Transient Data Controls** panel, set the current **TIME STEP** to **32** and click **Apply**.
3. Bring up the **Boundary Surface** panel, make surface **#4** current and turn **Visibility** off.
4. “Quick-pick” the Exhaust coordinate plane and change the value of **Z** from **17.5** to **10**. “Quick-pick” the Inlet coordinate plane and change the value of **Z** from **1.0** to **9**. [\[Page 75 User's Guide\]](#)
5. To see the variation in the gasoline concentration more clearly, use the “CFD Calculator” as before to create a new variable, **log(“gasoline”+1e-06)**. Click on **Scalar...**, then on the **Function Selection** panel, select **log(“gasoline”+1e-06)**. Change the **COLORING** from **Geometric** to **Scalar**. Turn **Show Mesh** off. On the **Colormap** tab, change the range to a minimum of **-7** and a maximum of **-3.25**.
6. On the main toolbar, click **Unify** to apply the same settings to the other coordinate surface.
7. To change the labels for these surfaces, “quick-pick” each label. On the **Annotation** panel, enter the appropriate text directly into the **Edit Text:** field.

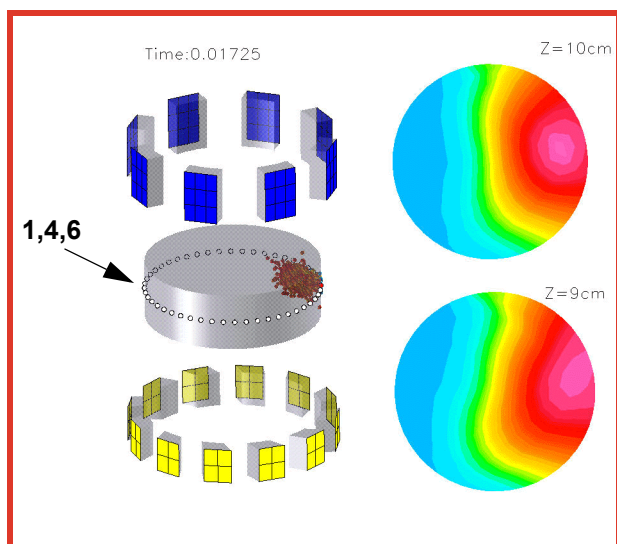


8. On the **Annotation** panel, click **Create Text**. On the **Annotation Create** panel, enter **Crank Angle = %%ALPHA** to display the crank angle the KIVA dataset has stored as a constant for each time step.
9. Bring up the **Particle Paths** panel and **Import ex4p.fvp** to read in the file containing the trajectories of individual massed particles obtained from the otape9 file and converted into a **FieldView** format for this problem.
10. Change **COLORING** from **Geometric** to **Scalar**. Particles can be colored by any scalar value that they have been saved with. For **Scalar Variable**, click **Select...** and select **Temperature**. Bring up **View** → **Rendering Options** and turn **Presentation Quality** on to show the particles as shaded spheres. [\[Page 360 Reference Manual\]](#)
11. Reopen the **Transient Data Controls** panel to animate the sequence. Set **Begin** and current **TIME STEP** to **2** and click **Apply**. Choose **Tools** → **Flipbook Build Mode**. Click **OK** to dismiss the **Flipbook Display Size Warning** panel. On the **Transient Data Controls** panel, turn **Build** on. When the building is complete, the **Flipbook Controls** panel may be used to play or **Save** the flipbook.
12. “Quick-pick” the cylinder boundary surface. On the **Boundary Surface** panel, change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading, **Transparency** from **0** to **50%** and **Geometric COLORING** to gray. (The figure shown at the top right of the previous page is for a **TIME STEP** value of **31**).
13. Follow the instructions in Step 11 to create an animation of this sequence.

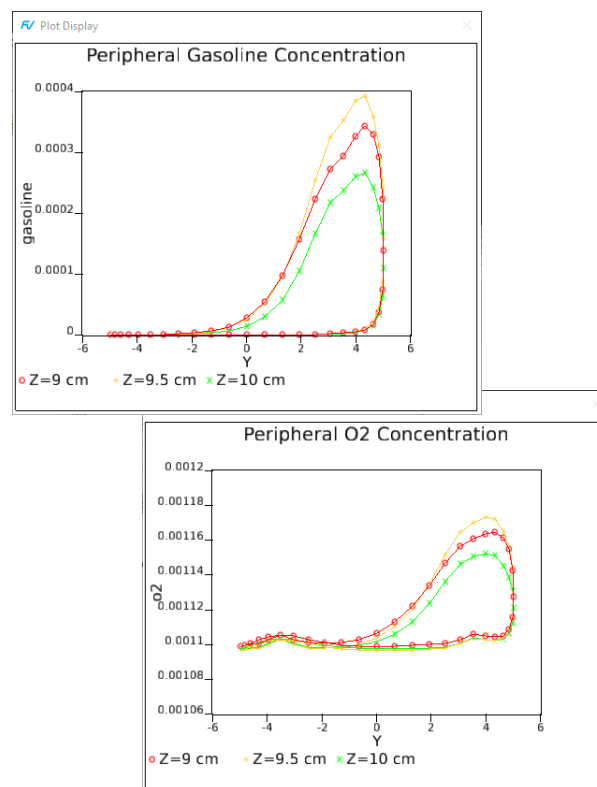
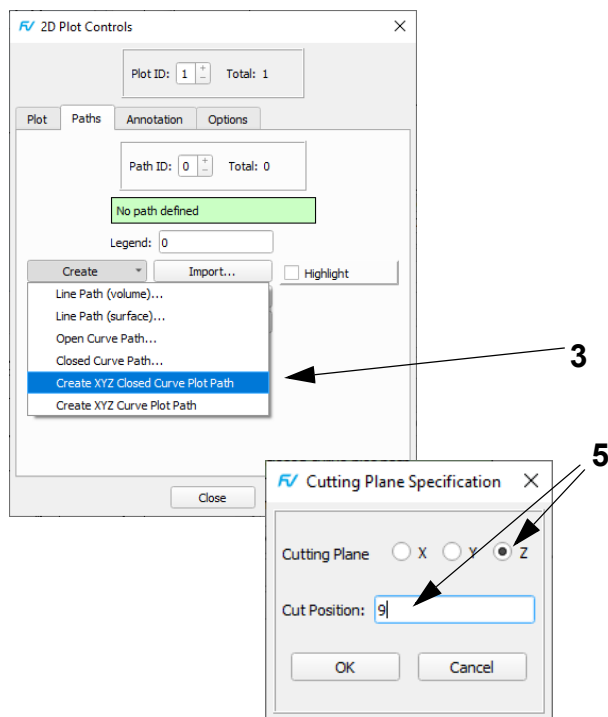


Step 4: 2D Plots of Gasoline and O2 Concentration

Here we show how to use 2D plots to get the range of various scalars around the cylinder periphery near the spray injection point.



1. On the **Transient Data Controls** panel, set the **TIME STEP** to 34. "Quick-pick" the boundary surface. Bring up the **Function Specification** panel and load the **Scalar** register with **gasoline**.
2. Click the side toolbar **Plot** icon or navigate to **Visualization Panels** → **2D Plots...** to bring up the **2D Plot Controls** panel. On the **Plot Tab**, click **Create**. [Page 270 Working with FieldView]
3. On the **Paths** tab, click **Create** and choose **Create XYZ Closed Curve Plot Path** from the pulldown menu.
4. Position the mouse near the middle of the cylinder and <Ctrl + left click> to define the 2D plot location. The exact location will be adjusted in the next step.
5. On the **Cutting Plane Specification** panel, change the **Cutting Plane** axis to **Z**. For **Cut Position**, enter **9.0** to position the plane exactly. Click **OK** and **Close** the panel.
6. On the **2D Plot Controls** panel **Plot** tab, turn **Show Path** on, and Set the **Horizontal Axis/Plotting Direction** to **Y**. Click **Axes...** On the **Horiz Axis** tab, change **TICK MARKS Major Unit**: from 1.2 to 2. (Not shown. For the next few steps, please refer to the panels shown on the next page).
7. To create the figures shown to the right, repeat steps 3 through 5, and position the planes at **Z=9.5** and **Z=10** for the second and third paths, and continue to next step.

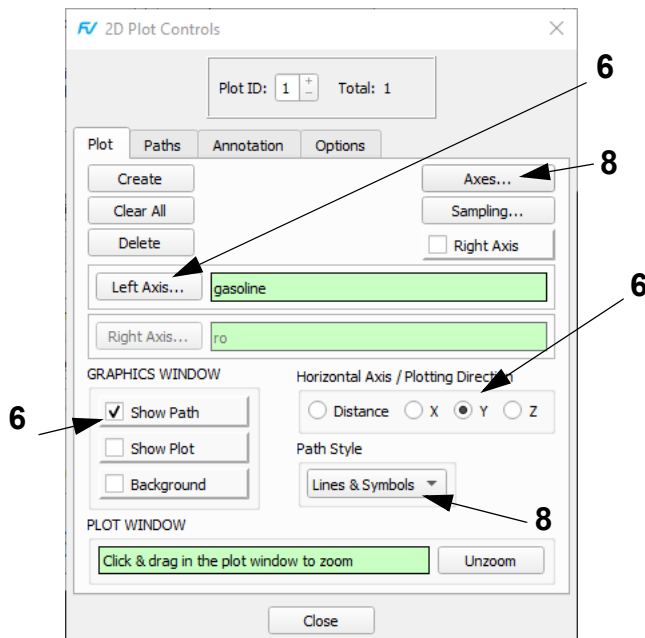
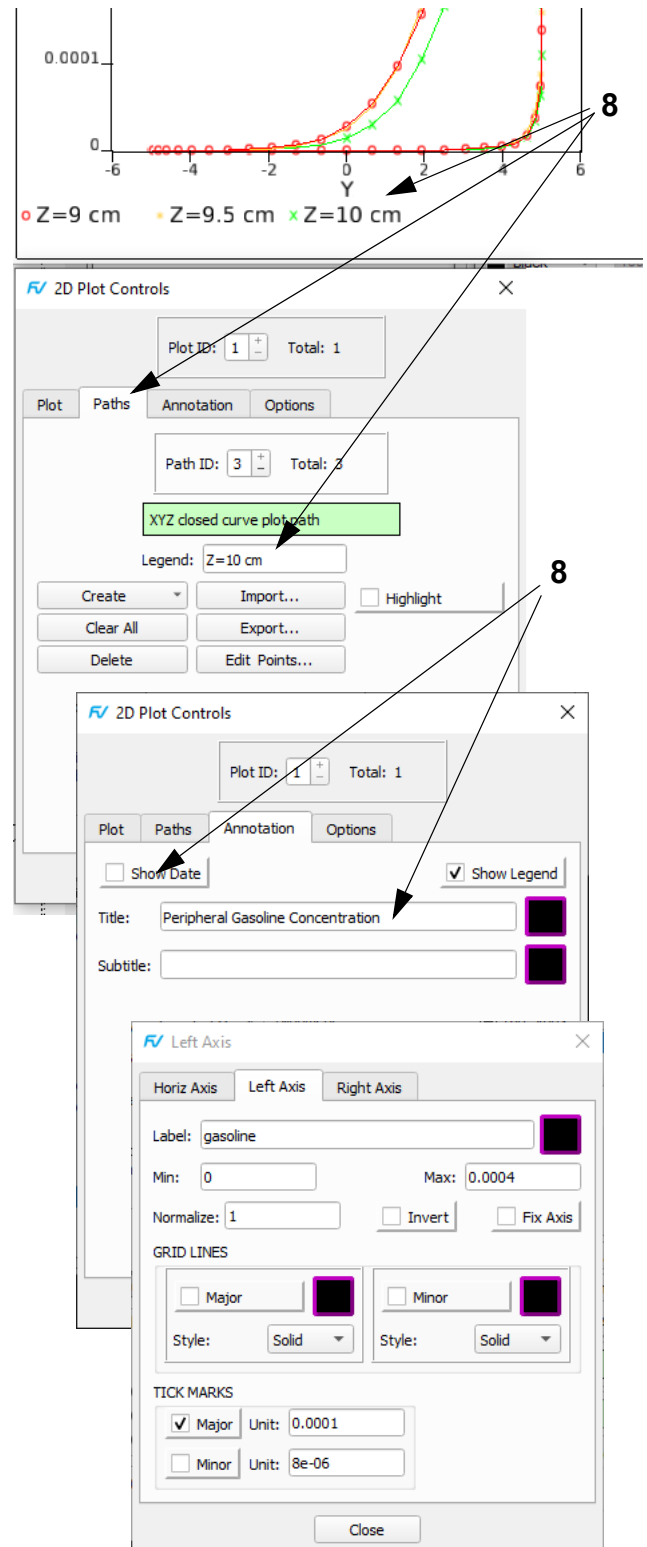


8. On the **Annotation** tab, turn **Show Date** off. Note that two quantiles will be plotted, and that the 2DPlot Panel settings here correspond only to the **Gasoline** plot.

For the plotting **Gasoline**, enter **Peripheral Gasoline Concentration** in the **Title:** field of the **Annotation** tab. On the **Plot** tab, change the **Path Style** from **Lines** to **Lines & Symbols**. On the **Paths** tab, change each **Legend:** from **Path ID (1, 2, and 3)** to show the cut position (**Z=9, 9.5, and 10 cm**). On the **Left Axis** tab, change **TICK MARKS Major Unit:** from 4e-005 to 0.0001. On the **Left Axis** tab, change **TICK MARKS Major Unit:** from 4e-005 to 0.0001.

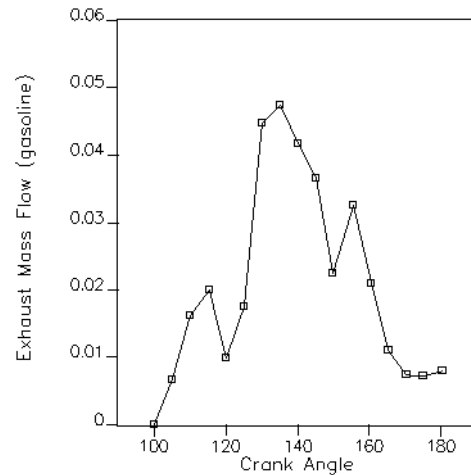
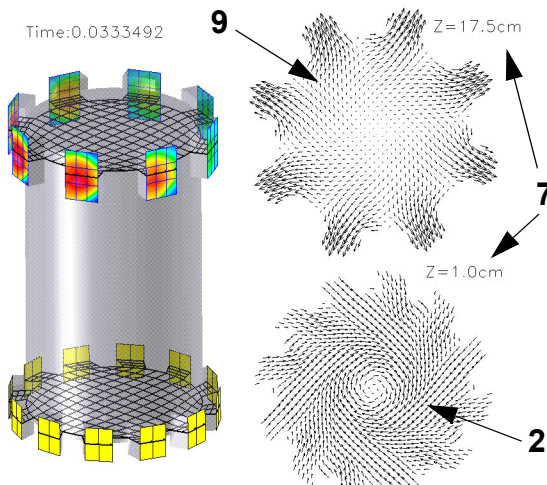
For plotting **O2**, Click on **Left Axis...** then, on the **Function Selection** panel, select **o2** and click **Calculate** to change the function plotted on the vertical axis. Enter **Peripheral O2 Concentration** in the **Title:** field of the **Annotation** tab. On the **Left Axis** tab, change **TICK MARKS Major Unit:** from 4e-005 to 2e-005.

9. You can examine how these curves change with time by changing the current **TIME STEP** on the **Transient Data Controls** panel.
10. When you are finished, **Close** the **2D Plot Controls** panel.

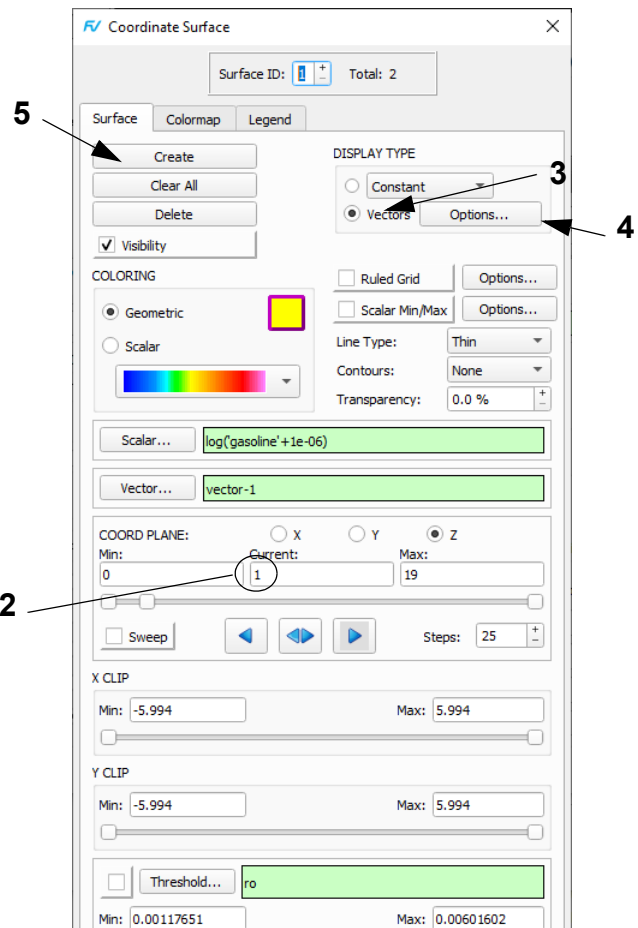


Step 5: Display the Exhaust Mass Flow as a Function of Crank Angles

Lastly, we show how to use the function calculator to create a new function which corresponds to the Exhaust Mass Flow of gasoline. A script is used to show how this calculation can be applied for several time steps.



1. On the **Transient Data Control** panel, set the **TIME STEP** to **65**.
2. “Quick-pick” the **Z=10** cm coordinate plane and change the value of **Z** from **10** to **17.5**. “Quick-Pick” the **Z=9** cm coordinate plane and change the value of **Z** from **9** to **1**.
3. Change the **COLORING** from **Scalar** to **Geometric** and the **Geometric** color to black. Change the **DISPLAY TYPE** to **Vectors**.
4. Click **Options...** On the **Vector Options** panel, change the **Vector Scale** to **2** and the **TYPE** to **Projected**. Turn **Uniform Sampling** on, set the **Number of Samples** for **X** and **Y** to **40** and click **OK**. On the main toolbar, click **Unify** to make the other surface match. “Quick-pick” the **Z=17.5** surface, bring up the **Vector Options** panel and change the **Vector Scale** from **2** to **8** to make the vectors on this surface more visible.
5. Click **Create** to copy the current **Z=17.5** surface in its original location without transformation. Change the **DISPLAY TYPE** from **Vectors** to **Constant**, then to **Mesh**. Change the **Line Type** from **Thin** to **Medium**.
6. “Quick-pick” the **Z=1.0** surface to make it current and **Create** a copy. Change the **DISPLAY TYPE** and **Line Type** as done previously. (Note: Do not use **Unify** for this, as it would change the attributes of all coordinate surfaces including the vector ones, which we do not want to alter.)



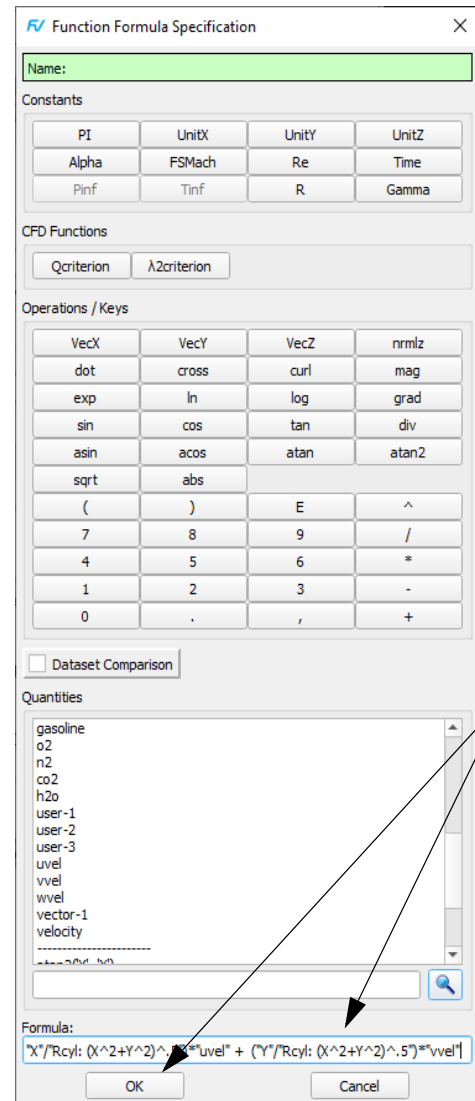
7. "Quick-pick" the **Z = 10cm** label. On the **Annotation ...** panel, edit the text to reflect the current **Z** value, **17.5**. Similarly, "quick-pick" the **Z = 9cm** label and edit it to **Z = 1.0cm**. **Close** the **Annotation** panel.
8. Click the main toolbar **f_x** icon. On the **Function Specification** panel, click **Create...** On the **Function Formula Specification** panel ("CFD Calculator"), enter **Formula: ("X"/"Rcyl: (X^2+Y^2)^.5")*"uvel" + ("Y"/"Rcyl: (X^2+Y^2)^.5")*"vvel"** for radial velocity $U_r = [(x/R)U_x + (y/R)U_y]$ and name the function **Uradial**. Create a second function equal to **Uradial * gasoline**.
9. Bring up the **Boundary Surface** panel and make the **Outlet** the current surface. Change the **Scalar Function** to **Uradial*gasoline** and **COLORING** from **Geometric** to **Scalar** (panel not shown).
10. Click the side toolbar **Integrate** icon or navigate to **Tools** → **Integration...** to bring up the **Integration Controls** panel. Change the **Integration Mode** to **Current Surface**. Click **Integrate** to get a value for the mass flow rate of gasoline (see next page).
11. Set the current **TIME STEP** to **58** and **Save a Complete Restart** named **massflow**. A script can be used to obtain a series of results for the exhaust gasoline mass flow for more than one time step. A sample script named **calc_mass.scr** is included with this tutorial. A partial listing appears below:

```

RESTART ALL massflow
SELECT BOUNDARY 3
INTEGRATE current masstime2
TIME STEP 1 58
SELECT BOUNDARY 3
INTEGRATE current masstime2
TIME STEP 1 59
SELECT BOUNDARY 3

```

12. The script reads the restart to begin at TIME STEP 58. The **SELECT BOUNDARY** command makes Boundary Surface ID: 3, which corresponds to the exhaust outlet, current. The next command performs the integration and sends the result to a file named **mass-time2**. The sequence is repeated for each time step. If Boundary Surface ID: 3 is BOUNDARY TYPE Outlet, navigate to **File** → **Open Restart** → **Script...** and **Open calc_mass.scr**. If not, edit **calc_mass.scr** in a text editor to match the Boundary Surface ID: number to the **Outlet** before opening the script in **FieldView**.

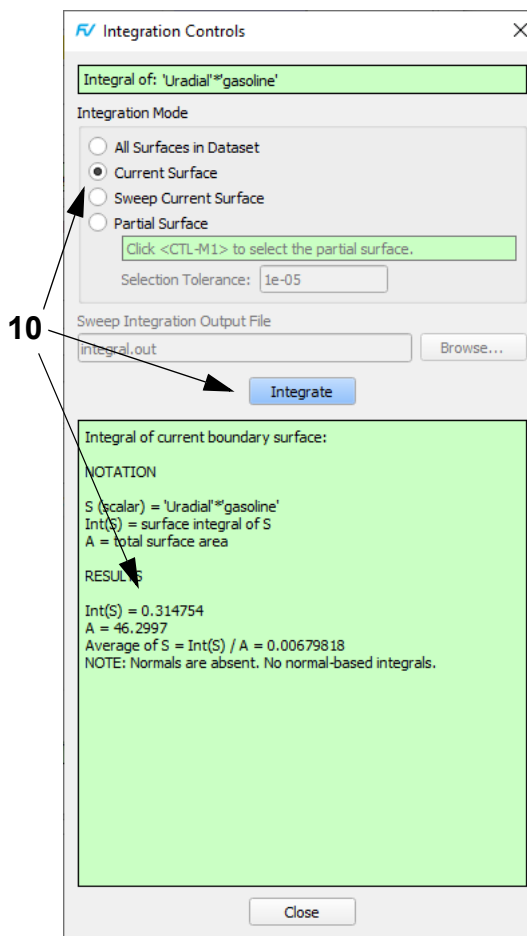


8

13. The results saved in the **masstime2** file can be combined with information from the KIVA **otape12** file about the **crank** angles for these time steps to produce the plot shown in the figure to the upper right. A new file can then be created with any text editor by typing in the **crank** angles from the **otape12** file and the **Average Value** integration values from the **masstime** file. This new file can be written in the **FieldView 2D Plot** file format for import and plotting. This format assumes X, Y, and Z location values, so the **crank** angle is entered for one of these (Z, in the example below) and the others are set at **0.0**. An example of what the import file might look like follows:

```
VARIABLES="X","Y","Z","massflow"
ZONE J=1
0 0 100.126 1.23E-07
0 0 105.204 .00663701
0 0 110.144 .0161613
0 0 115.32 .0199823
0 0 120.22 .00984395
0 0 125.05 .0175579
0 0 130.152 .0447927
0 0 135 .0473818
0 0 140.152 .0417616
0 0 145.272 .0364801
0 0 150.0158 .0225386
0 0 155.431 .0325474
0 0 160.46 .0208706
0 0 165.307 .0111051
0 0 170.251 .00743575
0 0 175.119 .00723032
0 0 180.195 .00800244
```

14. Change the **Horizontal Axis/Plotting Direction** to **Z**. The x- and y-axis titles can be changed by pressing the **Axes** button and editing the **Label** field under the **Horizontal Axis** and **Left Axis** tabs.



Chapter 13

Feature Extraction Tutorial

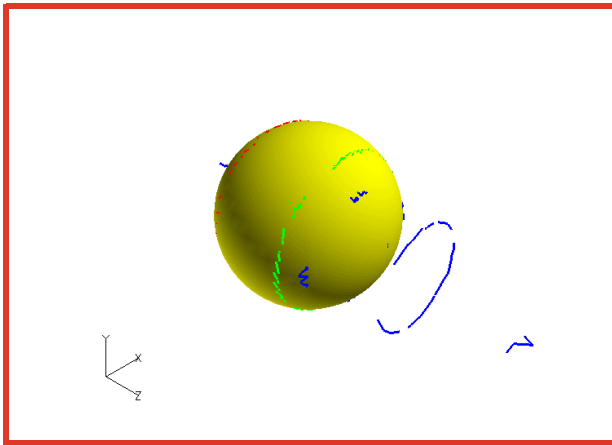
13

In this tutorial, we will examine flow around and within a water droplet. The 'interface' boundary (droplet) separates the water and air phases, and is immiscible; that is there is no phase transfer. In the first step we will create and orient the droplet geometry. In the next few steps, we will be making use of the Vortex Core and Separation/Reattachment Feature Detection tools in **FieldView**. In this step we display the droplet as a Boundary Surface and visualize the Vortex Cores and Separation/Reattachment Lines.

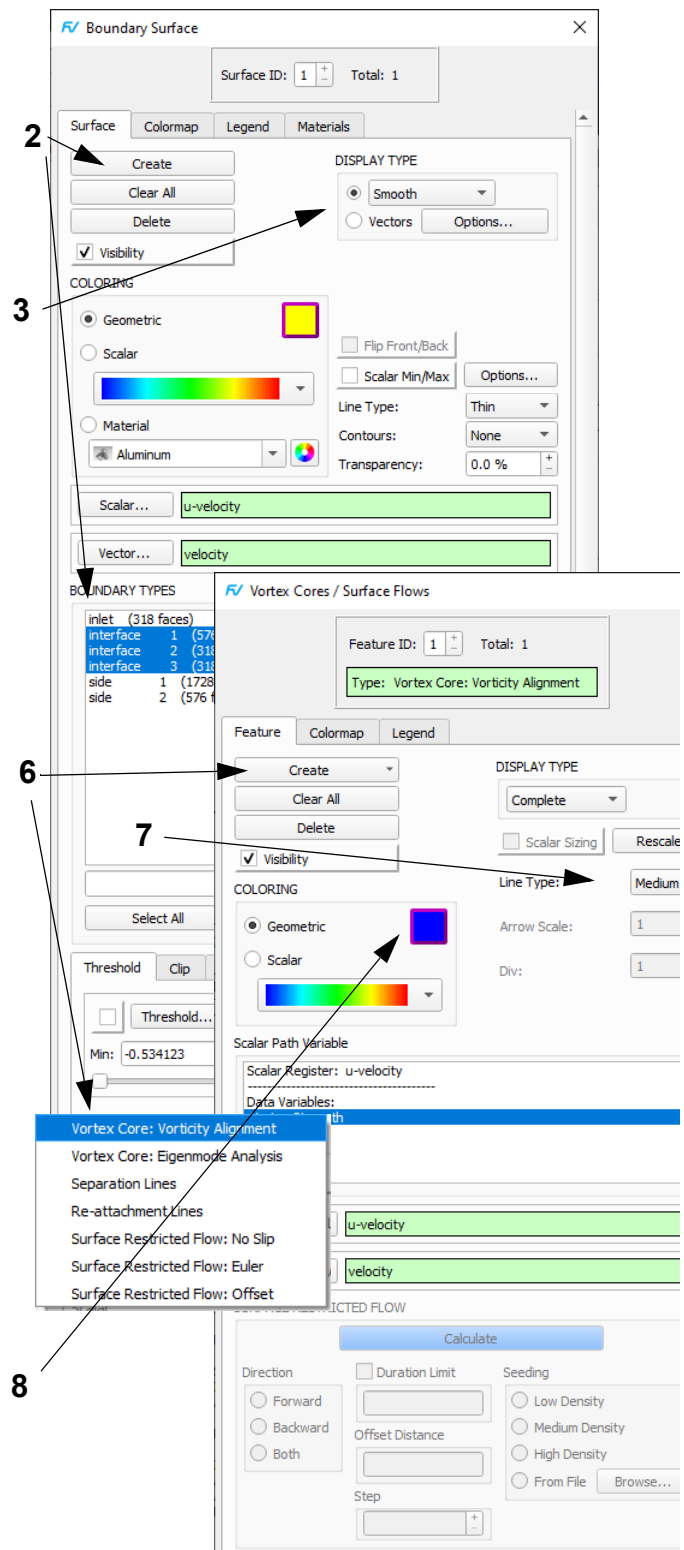
What You Will Learn

- [How to automatically display Vortex Cores](#)
- [How to automatically display Surface Streamlines](#)
- [How to analyze two datasets simultaneously](#)
- [How to use the Vortex Cores for Streamline Seeding](#)
- [How to show internal flow structures](#)
- [Create a keyframe animation for all vortex cores](#)

Step 1: Create Vortex Cores

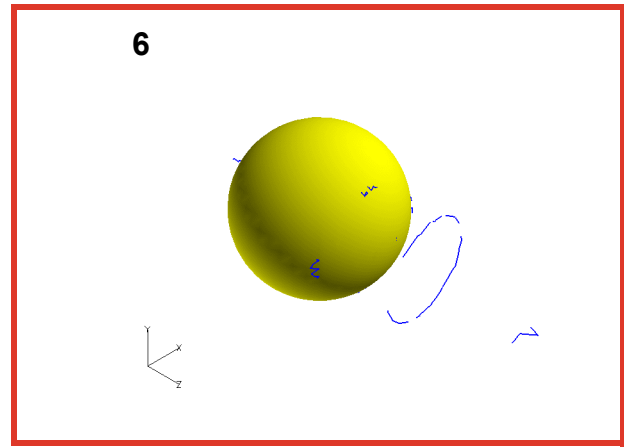
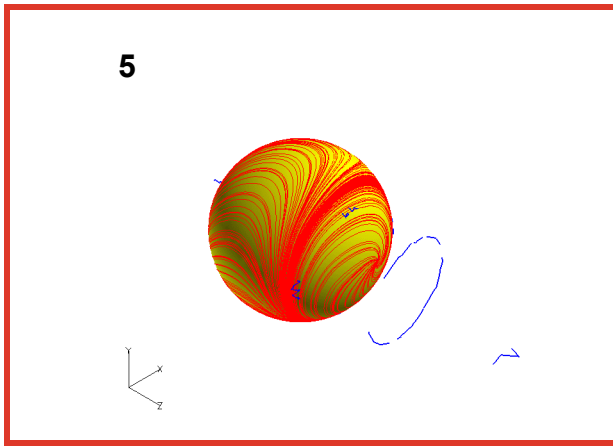


1. Bring up the **File** → **Data Input** → **AcuSolve [FV-UNS Export]** panel, click **Read Grid or Combined Data...** and **Open** the **FieldView** Unstructured dataset **recirc.uns**. Close the **AcuSolve [FV-UNS Export]** panel. [\[Page 36 Reference Manual\]](#)
2. To visualize the geometry, bring up the **Boundary Surface** panel. Click **Create**. Select **interface 1**, **interface 2**, and **interface 3** and click **OK**. [\[Page 240 Working with FieldView\]](#)
3. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
4. Change the **Geometric** color chip to yellow.
5. On the Transform Controls toolbar, turn the **Outline** off. On the Transform Controls toolbar, click the **Center** button.
6. From the **Visualization Panels** pull-down, bring up the **Vortex Cores / Surface Flows** panel. Click **Create** and select **Vortex Core: Vorticity Alignment** to create vortex cores using the Vorticity Alignment method. [\[Page 251 Working with FieldView\]](#)
7. Change the **Line Type**: from **Thin** to **Medium**.
8. Change the **Geometric** color chip to blue.

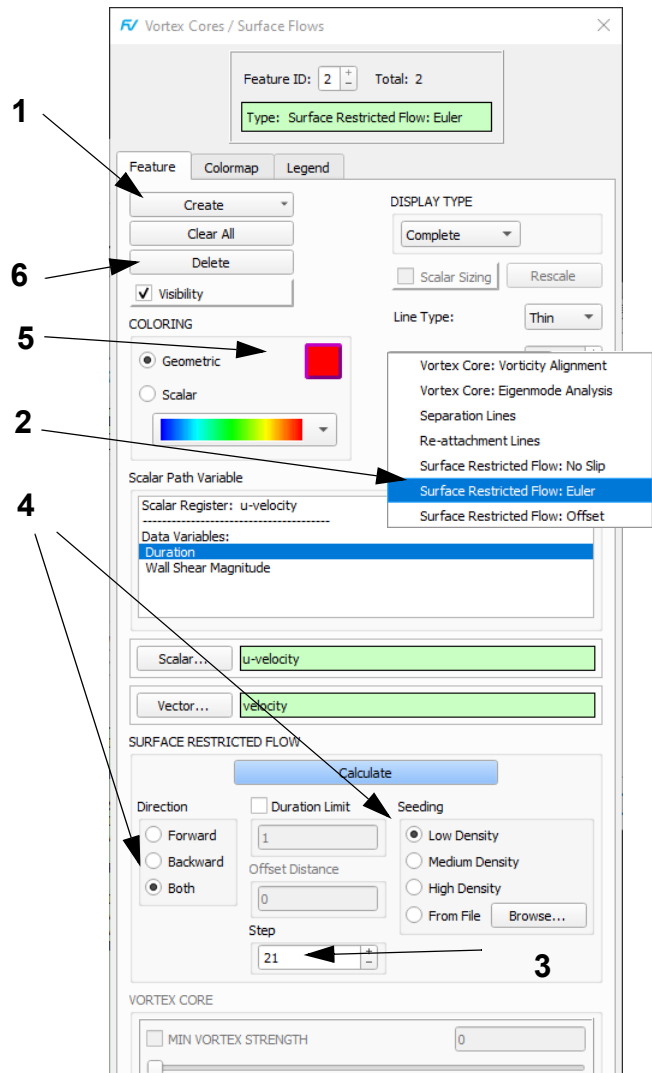


Step 2: View Surface Flows

In this step, we will show surface flow patterns on the droplet using an Euler assumption by creating a new surface flow feature.

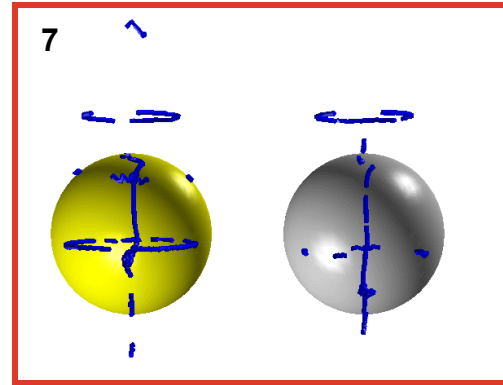
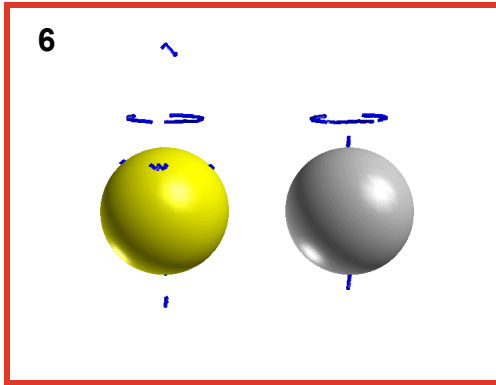


1. On the **Vortex Cores / Surface Flows** panel, click **Create**.
2. In the pull-down menu, select **Surface Restricted Flow: Euler**.
3. For the number of steps, use the default **Step** value of 21.
4. Ensure that **Direction** is set to **Both** and that **Seed Density** is set to **Low**. Then press **Calculate**.
5. Change the **Geometric** color chip to red.
6. Click **Delete** to delete the surface feature.

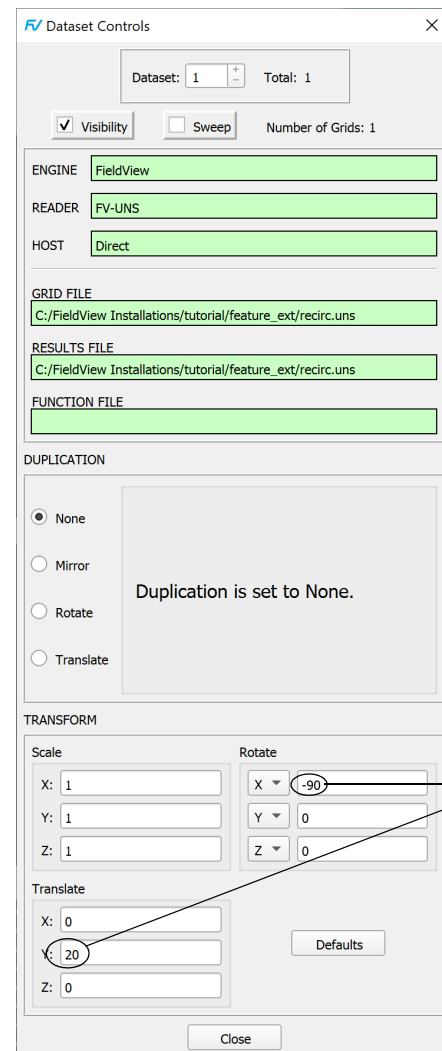


Step 3: Analyze Two Datasets Simultaneously

In this step we **Append** the current dataset with a duplicate copy. We then transform the second dataset and display Vortex Cores using the Eigenmode Analysis method.

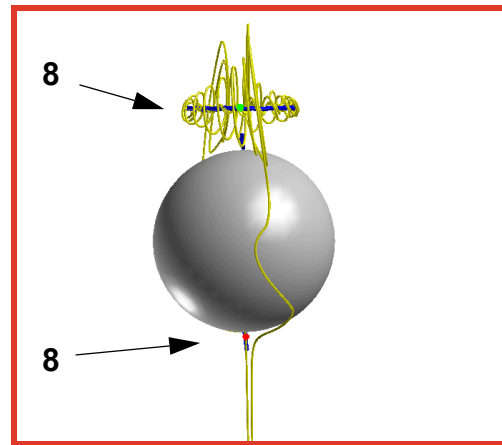
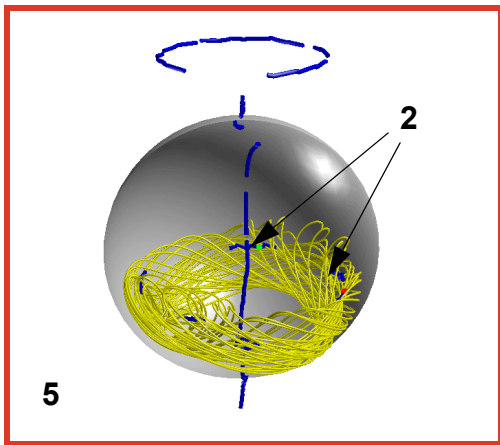


1. Bring up the **File** → **Data Input** → **FV-UNS** panel, change **INPUT MODE** from **Replace** to **Append** and re-read the dataset.
2. While the 2nd dataset is current, bring up the **Boundary Surface** panel and **Create** the same boundary surface as before, selecting **interface 1**, **interface 2**, and **interface 3** and clicking **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth Shading**. Change the **Geom** color to gray.
3. On the Transform Controls toolbar, click the icon to bring up the **Defined Views** panel. Choose **-X**, then **Close** the panel. From the **View** pulldown menu, turn off the **Axis Markers**. Click the **Dataset...** button on the main toolbar to bring up the **Dataset Controls** panel. Set an **X ROTATE** value of **-90** degrees. Switch to the 1st dataset and also set an **X ROTATE** value of **-90** degrees, then set the **Y TRANSLATE** value to **20**. This will translate the 1st (yellow) droplet to the left. **Close** the panel. Now **Center** the view.
4. “Quick-Pick” (double-click) the 2nd (gray) dataset. Bring up the **Vortex Cores / Surface Flows** panel and click **Create**. In the **Create Feature** pull-down menu, select **Vortex Core: Eigenmode Analysis**.
5. Change the **Line Type:** to **Medium**. Change the **Geometric** color to blue.
6. From the **View** pulldown menu, turn on the **Presentation Quality**. On the **Vortex Cores / Surface Flows** panel, click **Rescale**.
7. For each dataset, “Quick-Pick” on the Boundary Surface. On the **Boundary Surface** panel, change the threshold function to **X**, change **Min** to **0.0** and turn on the **Threshold Clip**

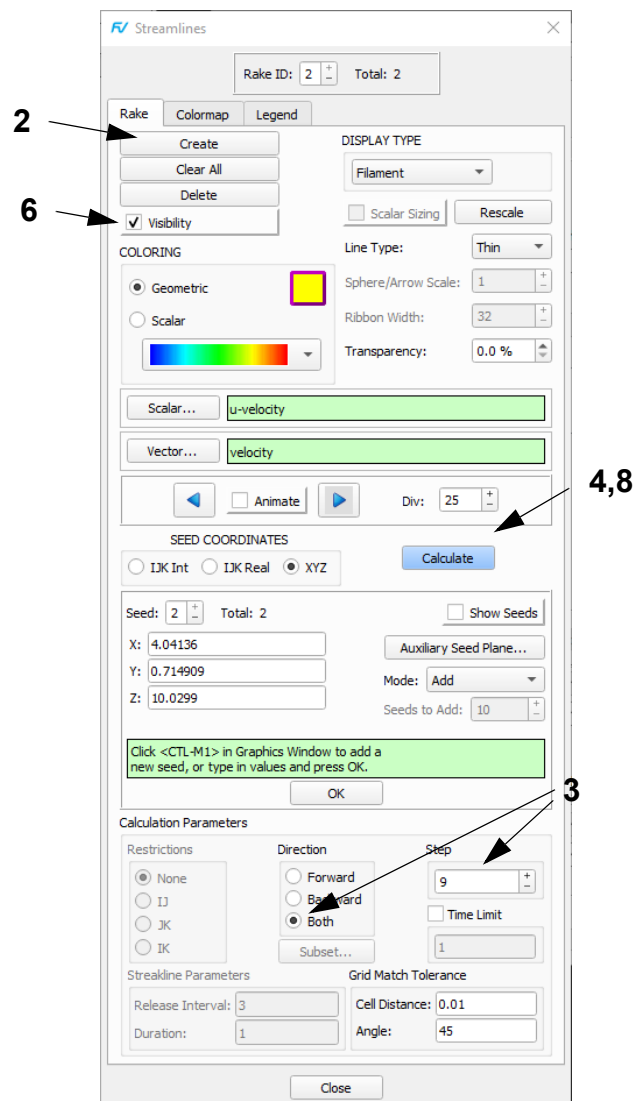


Step 4: Show Internal Flow Structure

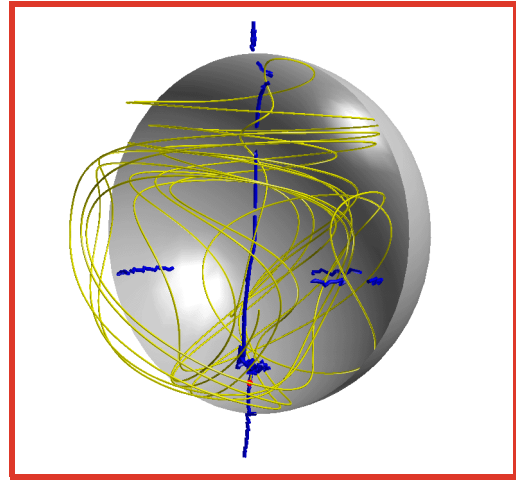
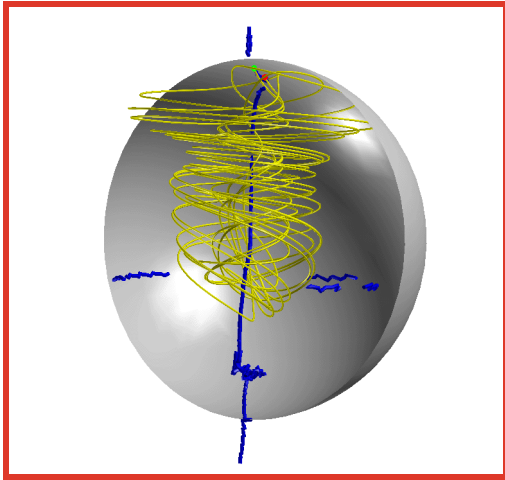
In this step we use streamline seeding directly on the vortex cores to show the flow structures.



1. From this step on, we will only use the dataset showing the Eigenmode Analysis vortex cores. Bring up the **Dataset Controls** panel, change to Dataset #1 (if necessary) and turn off the **Visibility**. [Also turn off Dataset #1's **Visibility** on the **Vortex Cores / Surface Flows** and **Boundary Surface** panels.] On the **Dataset Controls** panel, switch to Dataset #2 and **Close** the panel. Using the mouse controls, zoom into the droplet and rotate the view down slightly to get a better view of the vortex segments inside the droplet.
2. Bring up the **Streamlines** panel. On the **Rake** tab, click **Create**. The default seeding mode is **Add** and the default coordinates are **XYZ**. Using the Control key and the left mouse button (**Ctrl-M1**), **Add only two seeds** directly on the blue ring vortex core, similar to those shown in the image above. [Page 184 Working with FieldView]
3. Change the **Calculation Parameters** as follows: Change the **Direction** from **Forward** to **Both**, and increase the **Step** number from **3** to **9** to increase the integration accuracy.
4. Press **Calculate**.
5. Change the **Geometric** color to yellow. Notice the complex streamline patterns inside the droplet.
6. Turn off the **Visibility** of the first streamline rake.
7. Using the mouse controls, zoom away from the droplet. Straighten up the droplet and rotate around to the non-thresholded side.
8. **Create** a second streamline rake. Again, using *only two seeds*, seed on the main outer ring and on the core at the bottom of the droplet. Press **Calculate**. Note the complex circulation pattern ringing the vortex core.

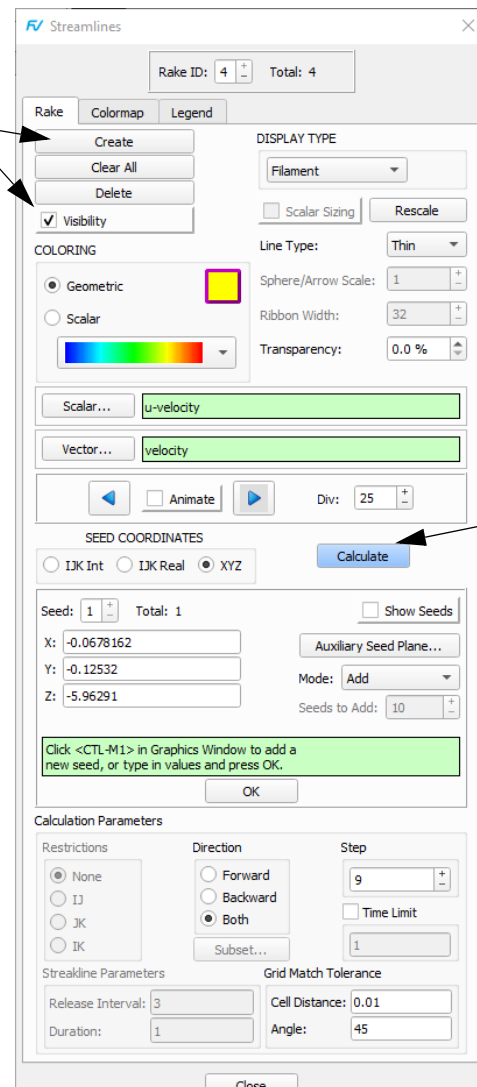


In this step we again use streamline seeding directly on the vortex cores to show internal flow structures.



1. Use the mouse controls to rotate the droplet back around to expose the interior. Zoom into the droplet, and rotate it a little down and around, as shown in the image above left.
2. Turn off **Visibility** for streamline rake #2. **Create** a third rake. **Add only two** seeds directly on the upper part of the vertical vortex core, close to the 'kinks', similar to those shown in the image above. Press **Calculate**. Notice the recirculation near the center of the droplet.
3. Turn off **Visibility** for the newly created third rake and **Create** a fourth. This time, **Add** a single seed near the bottom of the central vertical core to visualize the flow near the interface of the droplet. Press **Calculate**.

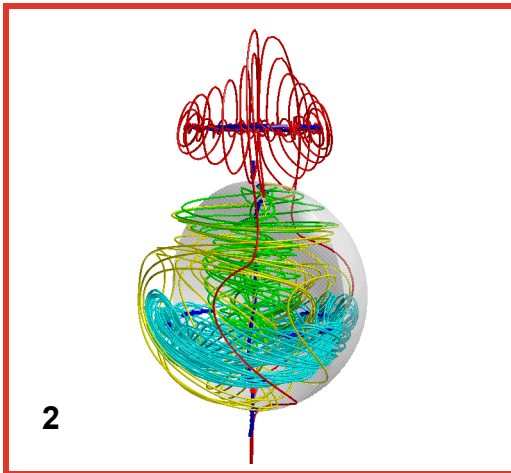
2,3



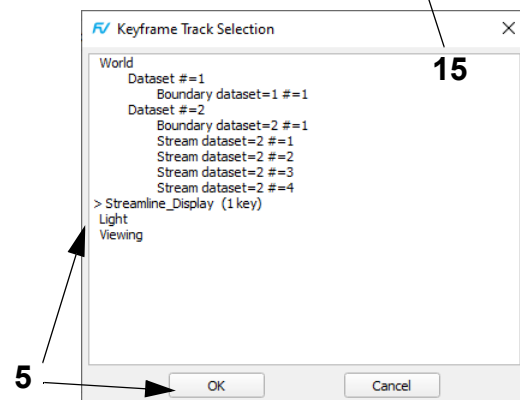
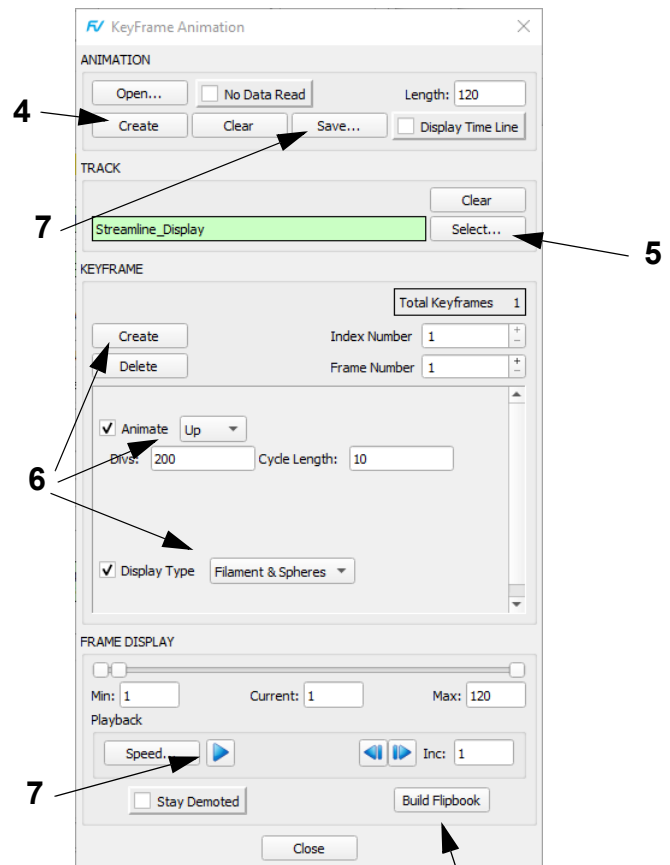
2,3

Step 5: Create a Keyframe Animation

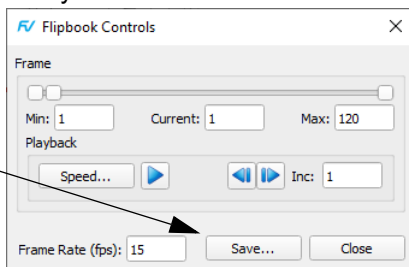
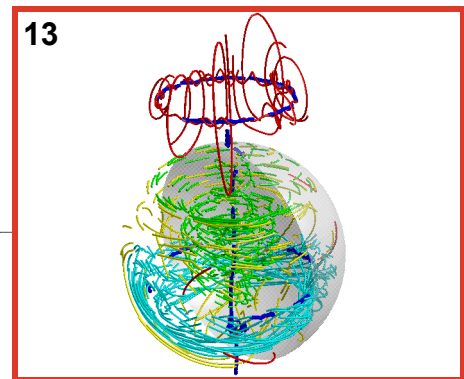
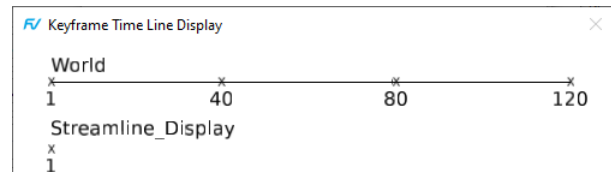
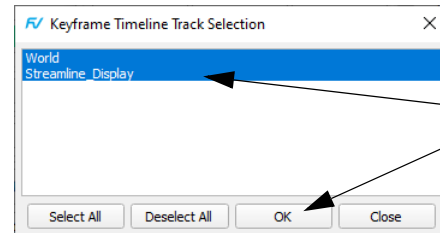
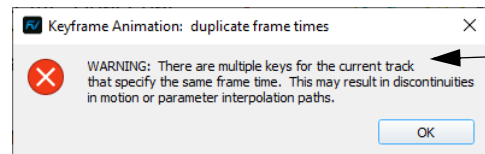
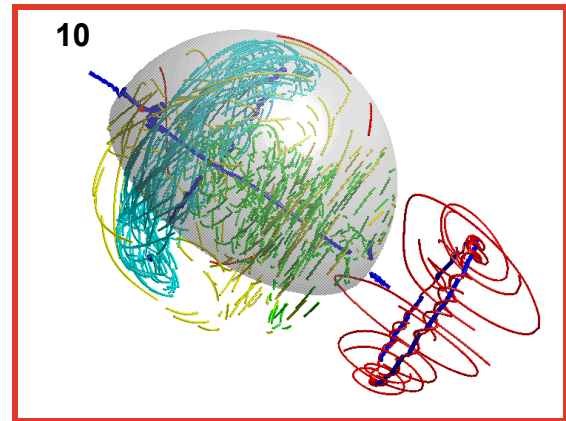
In this step we create a Keyframe Animation of Streamlines through all Vortex Cores.



1. Use the mouse controls to rotate the droplet down and slightly to the left, to expose a little more of the droplet surface. "Quick-pick" the droplet surface. On the **Boundary Surface** panel that comes up, set the **Transparency** value to 50%.
2. "Quick-pick" on the streamlines. On the **Streamlines** panel, make rake #1 current. Turn on **Visibility** and turn off **Show Seeds** for this rake. Change the **Geometric** color to light blue. For each of the other streamlines, turn the **Visibility** on and **Show Seeds** off. Make rake #2 red, rake #3 green and keep rake #4 yellow.
3. Use the **Tools** → **Graphics Layout Size** pull-down and choose **640x480 NTSC**. This will create an animation file of a standard size.
4. Bring up the **Keyframe Animation** panel from the **Tools** pulldown menu. Press the **ANIMATION Create** button to create a new keyframe animation. [\[Page 316 Reference Manual\]](#)
5. Press the **Select...** button. In the **Keyframe Track Selection** panel, select **Streamline_Display** and click **OK**.
6. Press the **KEYFRAME Create** button to create a keyframe at frame 1 for **Streamline_Display**. Change the **Animate** option from **Off** to **Up**. Change the value of **Divs** to **200** and the **Display Type** from **Complete** to **Filament & Spheres**. Note how changing **Animate** and **Display Type** automatically turns on the corresponding check box.
7. Press the Play button to preview the animation, then stop. Save a Keyframe Restart.



8. Click the **Select...** button. Choose **World** and click **OK**.
9. Use the slider to reset **FRAME DISPLAY** to frame 1. Press the **KEYFRAME Create** button to create a keyframe at frame 1 for the **World**. To keep the current view, turn on the **Transformation** button.
10. Change the **Current** frame value to 40. Click the **KEYFRAME Create** button. Interactively move the dataset using the mouse controls **M2** (middle mouse button) to rotate, **M3** (right mouse) to zoom, and the Spacebar toggle **M2** to Screen-Z rotate to create the view shown at right. This will also turn on the **Transformation** button.
11. We will now create a duplicate keyframe at the same frame number (40). This will help 'clamp' the view during the animation. Click the **KEYFRAME Create** button. You will get the pop-up shown to the right. Click **OK**. Turn on the **Transformation** button.
12. Change the **Current** frame value to 80. Click the **KEYFRAME Create** button and turn on the **Transformation** button. Again, **Create** a duplicate frame, this time at frame 80.
13. Now set the last keyframe, to provide the final view to interpolate to. Change the **Current** frame value to 120. Click the **KEYFRAME Create** button. Interactively move the World using the mouse controls to create the final view shown to the right. This will also turn on the **Transformation** button.
14. Turn on **Display Time Line**. This will bring up the **Keyframe Timeline Track Selection** panel. By default, all tracks with keyframes specified will be selected, so merely click **OK**. This will produce the **Keyframe Time Line Display** shown to the right. Turn off **Display Time Line** to dismiss the **Keyframe Time Line Display**.
15. Reset the animation to frame 1 and **Save** a final Keyframe Restart. From the **Tools** menu, set the **Graphics Layout Size** to 640x480 NTSC. Click **Build Flipbook** (see panel previous page) to create the animation file. Click **OK** to dismiss the **Flipbook Display Size Warning**. The **Flipbook Controls** panel can be used to Play and **Save** the animation.



Chapter 14

14

Heat Exchanger Tutorial

FieldView contains many features that allow engineers to compare designs by obtaining both analytical and visual results directly from 3D data. In this tutorial you will work on a heat exchanger by first learning how to display the data using cylindrical coordinates and how to make a theta coordinate surface of temperature distribution. You will then use these results and automatically create the same surfaces on a new heat exchanger dataset. In the remaining steps you will learn how to obtain and compare integration results from both datasets, create a 2D plot and then make a Key-Frame animation.

What You Will Learn

- [How to display cartesian data using cylindrical coordinates](#)
- [How to create a theta coordinate surface](#)
- [How to automatically create surfaces from previous datasets](#)
- [How to compare dataset results](#)
- [How to use partial Integration](#)
- [How to create 2-D plots from the intersection of surfaces](#)
- [How to create a KeyFrame animation from multiple datasets](#)

Step 1: Display with Cylindrical Coordinates



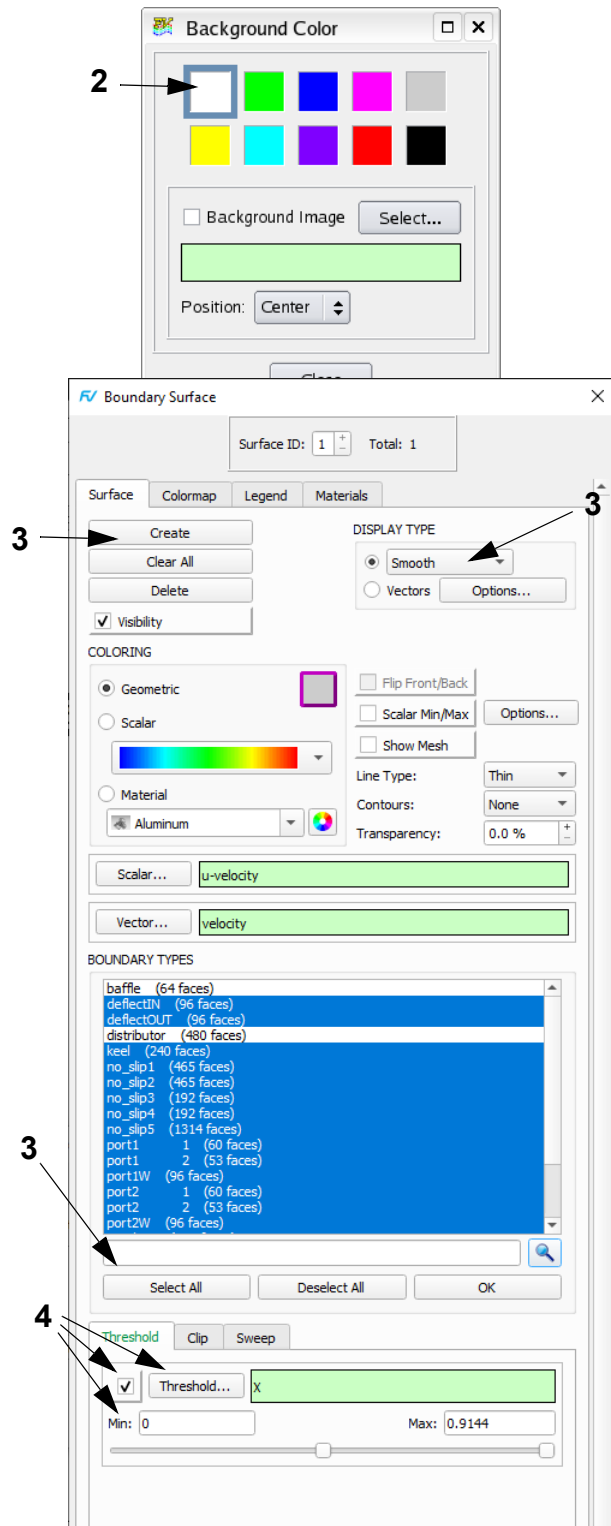
The pictures on each page correspond to the final image you should see when the steps are completed correctly.

1. Bring up the **File→Data Input→AcuSolve [FV-UNS Export]** Data reader. On the **AcuSolve [FV-UNS Export]** panel, choose **Read Grid or Combined Data...** From the file browser, choose the **cocurrent.uns** dataset and click **Open**. **Close** the **AcuSolve [FV-UNS Export]** panel. Note that when reading **cocurrent.uns** **FieldView** automatically finds the region file **cocurrent.uns.fvreg** that contains the following lines:

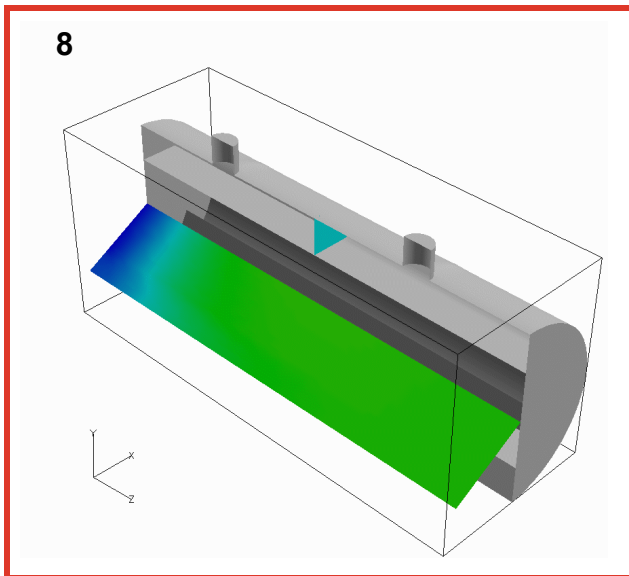
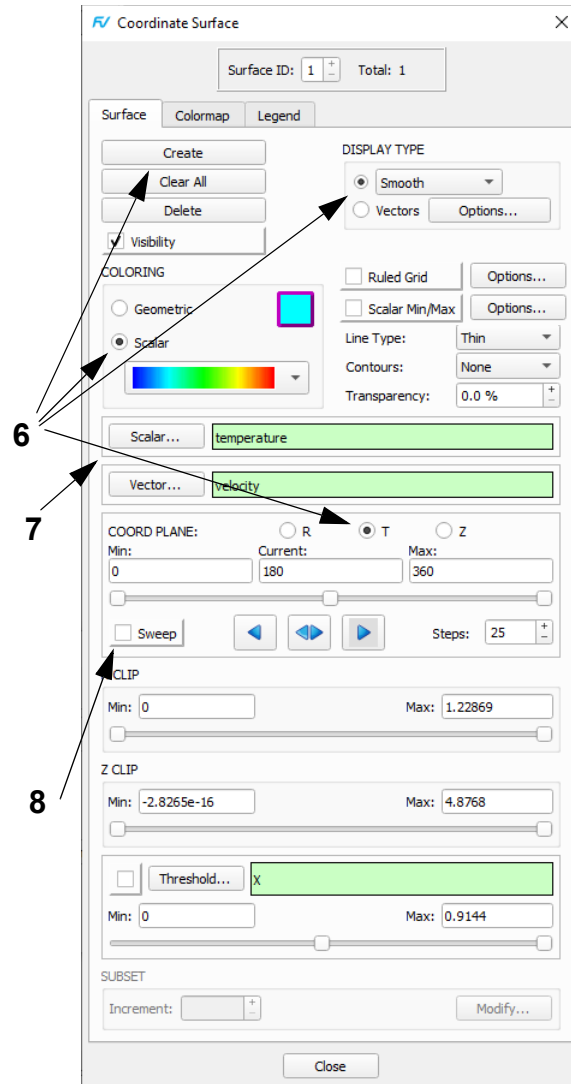
```
FVREG 1
DATASET_COORD_TYPE cylindrical
ORIGIN      0 0 0
Z_AXIS      0 0 1
THETA_AXIS  1 0 0
```

The region file tells **FieldView** to display the dataset in cylindrical rather than Cartesian coordinates. Some **FieldView** panels, such as the **Coordinate Surface** panel, will display 'R T Z' rather than 'X Y Z' coordinates. [\[Page 110 Reference Manual\]](#)

2. Bring up **View→Background Color...** (OR click the main toolbar icon to bring up the **Scalar Colormap Specification** panel and click **Background**). On the **Background Color** panel, click the white color chip to change the background color from black (default) to white.
3. Bring up **Visualization Panels → Boundary Surface** OR click the **Bound** icon on the side toolbar. On the **Boundary Surface** panel, click **Create**. **Select All BOUNDARY TYPES**, deselect (Ctrl + left click) **baffle** and **distributor**, and click **OK**. Set the **DISPLAY TYPE** to **Smooth** shading. [\[Page 240 Working with FieldView\]](#)
4. To cut away part of the heat exchanger, click **Threshold...** for **Threshold Function**. On the **Function Selection** panel, select **X** and click **Calculate**. Type in the value or move the left slider to change **Min** to 0. Turn **Threshold Clip** on.



5. **Create** another boundary surface, select **BOUNDARY TYPE** **baffle** and click **OK**. Note that **Surface ID** is now **2** with **DISPLAY TYPE** (**Smooth** shading) and **Geometric COLORING** (gray) inherited from **Surface 1**. Change the **COLORING** by clicking the color chip next to **Geometric**. On the **Geometry Color** panel, click the cyan (light blue) color chip and **Close** the panel.
6. Bring up **Visualization Panels** → **Coordinate Surface** OR click the **Coord** icon on the side toolbar. On the **Coordinate Surface** panel, click **Create**. Select **COORD PLANE: T** to display a theta plane at the default location of 180 degrees. Select **Scalar COLORING** and **DISPLAY TYPE: Smooth** shading.
7. To color the surface using temperature, click **Scalar...** for **Scalar Function**. Then on the **Function Selection** panel, select **temperature** and click **Calculate**.
8. Turn on **Sweep** to rotate the theta surface around the Z-axis.

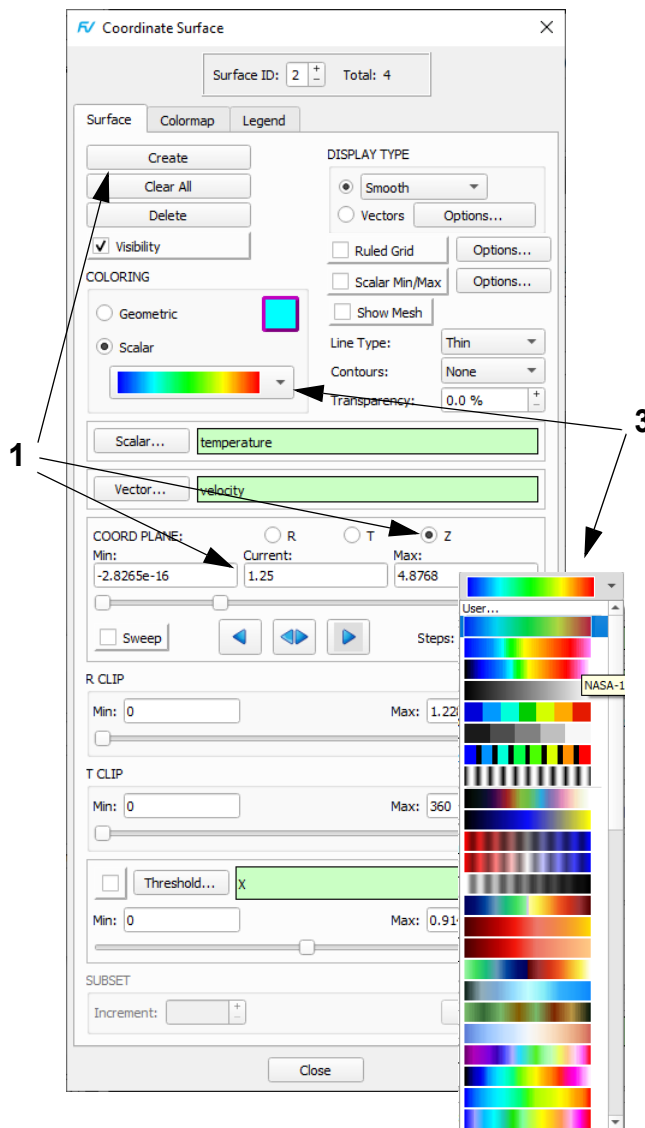


Step 2: Automatically re-create surfaces from a different dataset

In this exercise you will create a new set of surfaces on a dataset that was created in a previous dataset using a restart file.

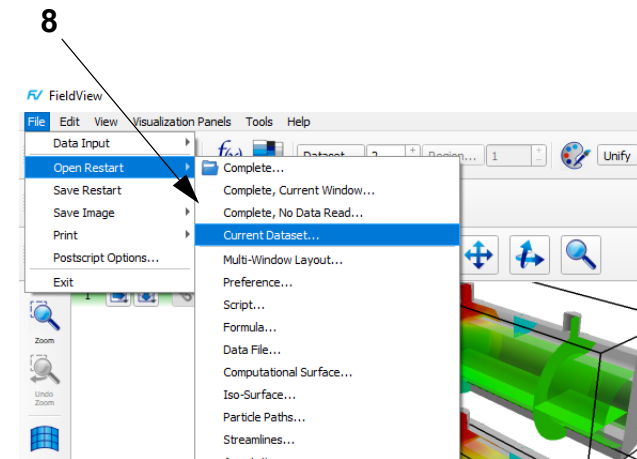
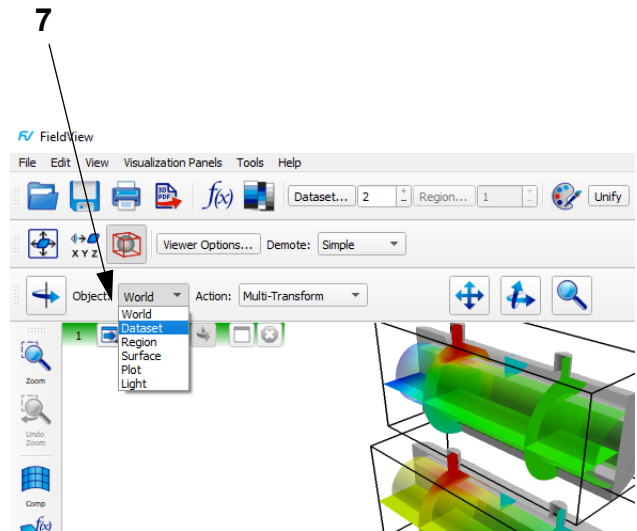
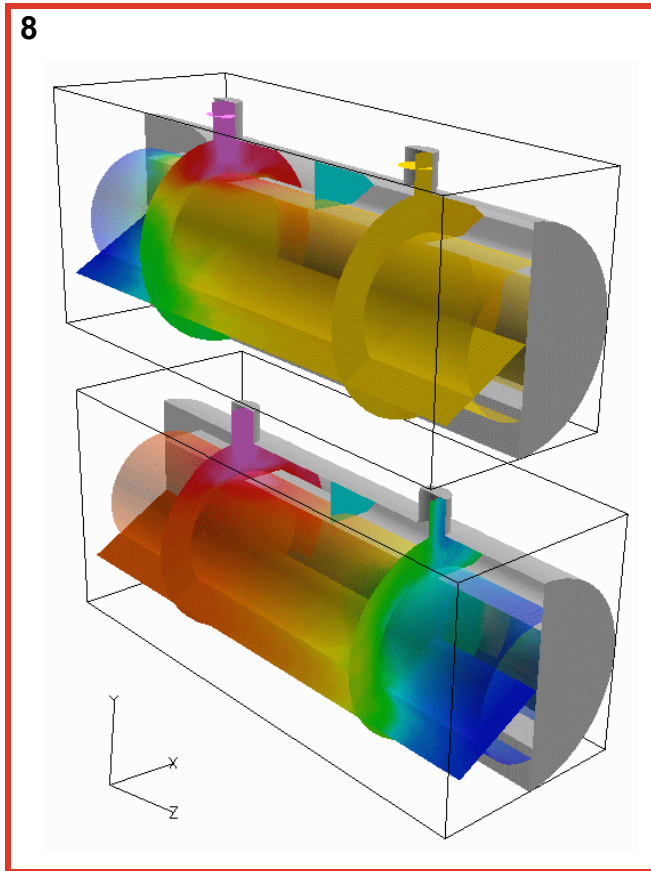
1. Navigate to **Visualization Panels** → **Coordinate Surface** or **quick pick** (double-click) the existing coordinate surface to bring up the **Coordinate Surface** panel. Click **Create** and set the **COORD PLANE** to **Z**. For the **Current** value, type in 1.25 and press Enter OR move the slider to 1.25. [\[Page 232 Working with FieldView\]](#)
2. Add two more coordinate surfaces at Z=3.75 and R=0.6. Note that a new coordinate surface inherits the properties of the existing one. Verify the following settings for all coordinate surfaces: **Scalar Function** = **temperature**, **COLORING** = **Scalar**, **DISPLAY TYPE** = **Smooth** shading. Any options that are different for any coordinate surface may be changed on the current coordinate surface and propagated to all others with **Tools** → **Unify**. [\[Page 345 Working with FieldView\]](#)
3. Click the colormap for direct access to the colormap selector pull-down, and select **NASA-1**. Then then propagate it to all coordinate surfaces with **Tools** → **Unify**, found on the main panel (not shown.)
4. **Quick-pick** (double-click) the R=0.6 **Coordinate Surface**. Change **Transparency** to 50% with the +/- keys.
5. Navigate to **File** → **Save Restart** → **Current Dataset...** to bring up the **SAVE RESTART: Current Dataset** browser. Type in the **File name: start_on_one** using the default **File type: Current Dataset (*.dat)** and click **Save**. [\[Page 260 Reference Manual\]](#)
6. Bring up the **File** → **Data Input** → **FV-UNS** data reader. On the **FV-UNS** panel, change the **INPUT MODE** to **Append** and choose **Read Grid or Combined Data...** From the file browser, choose the **counter.uns** dataset and click **Open**. As with the first heat exchanger dataset, the region file **counter.uns.fvreg** is automatically recognized when the dataset is read. **Close** the **FV-UNS** panel.

Note that after appending the second dataset, the number to the right of **Dataset...** on the main toolbar has changed from 1 to 2.



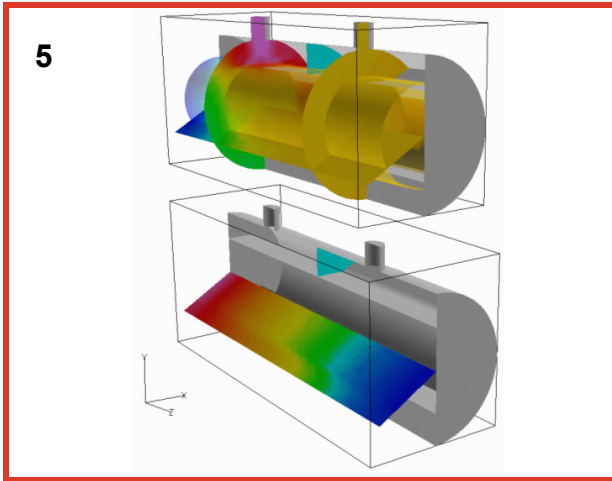
7. On the Viewer Toolbar, change **Object:** from **World** to **Dataset**. Left click and drag Dataset #2 (the current dataset) down to view both datasets simultaneously. (Change Object: back to World.)
8. Navigate to **File** → **Open Restart** → **Current Dataset...** to bring up the **OPEN RESTART: Current Dataset** browser. Select **start_on_one.dat** and click **Open**.

Note that reading a Current Dataset Restart does not reload datasets; only the duplication and region parameters and information on all surfaces are read from the Data File component restart (.dat). Thus surfaces created for the first dataset, **ccurrent.uns**, are automatically re-created for the second dataset, **counter.uns**, using the Current Dataset Restart saved in Step 5, **start_on_one.dat**.



Step 3: Compare Datasets

In this exercise we will create a function and compare the results on the two datasets simultaneously.



1. Click the main toolbar icon to bring up the **Function Specification** panel. Click **Create...** to bring up the **Function Formula Specification** panel. Scroll down the **Quantities** list and select **temperature**. Under **Operations/Keys**, click *****. From the **Quantities** list, select **velocity**. Note: At this point, **Dataset Comparison** should be off. Click **OK**. On the **Function Name Input** panel (not shown) click **Use Formula**. [Page 100 Reference Manual]
2. On the **Function Specification** panel, click **Create...** to bring up the **Function Formula Specification** panel. Scroll down the **Quantities** list to verify that the new '**temperature*velocity**' function formula is included. Turn on **Dataset Comparison**. Each function will now be prepended with its dataset number.
3. Create the following function formulas from the **Function Formula Specification** panel:

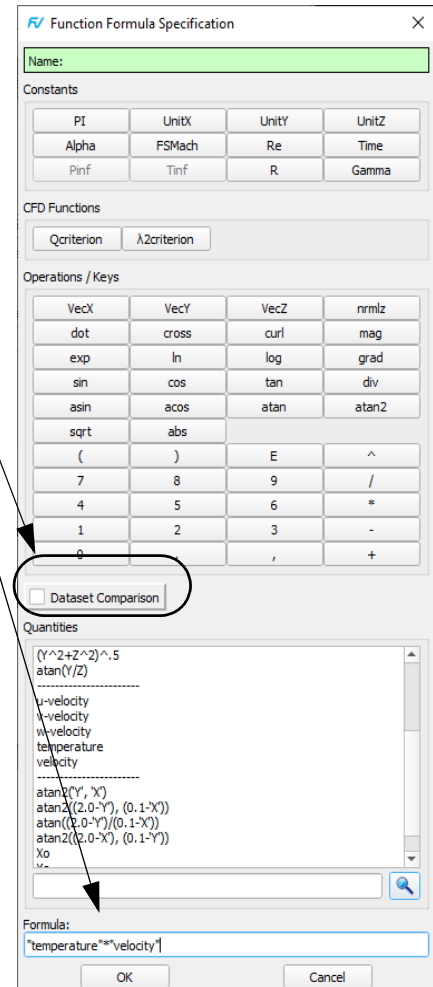
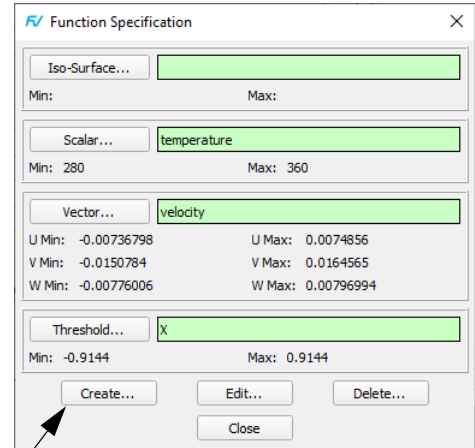
2:temperature – 1:temperature

2:temperature*velocity – 1:temperature*velocity

2:temperature

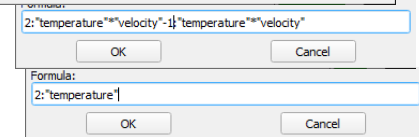
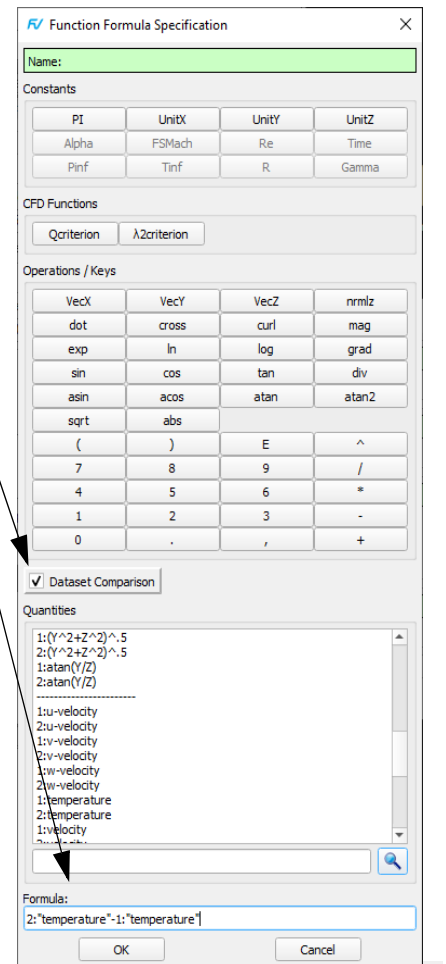
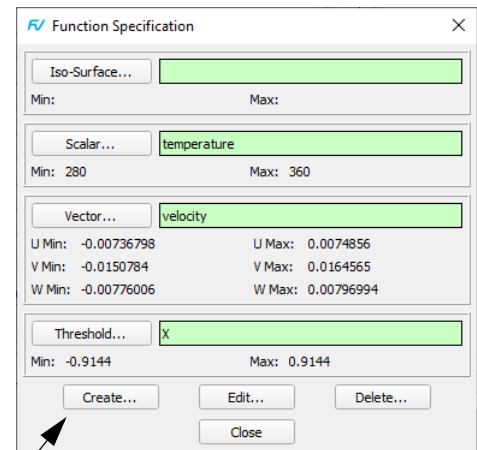
Note: By creating the function **2:temperature**, we can display the temperature from Dataset 2 on Dataset 1 directly. We will make use of this technique in one of the steps below.

4. Make sure that Dataset 2 is current (see number to the right of **Dataset...** on the main toolbar). Sequentially **quick pick** (double-click) each of the coordinate surfaces except the theta surface. On **Coordinate Surface** panel, turn off the **Visibility** of all coordinate surfaces except the theta surface.



5. Quick pick the theta surface. To the right of **Scalar Function**, click **Select...** On the **Function Selection** panel, select **2:temperature - 1:temperature** and click **Calculate**. On the **Coordinate Surface** panel, turn on **Sweep** to see the temperature difference between the two datasets (see illustration on preceding page).

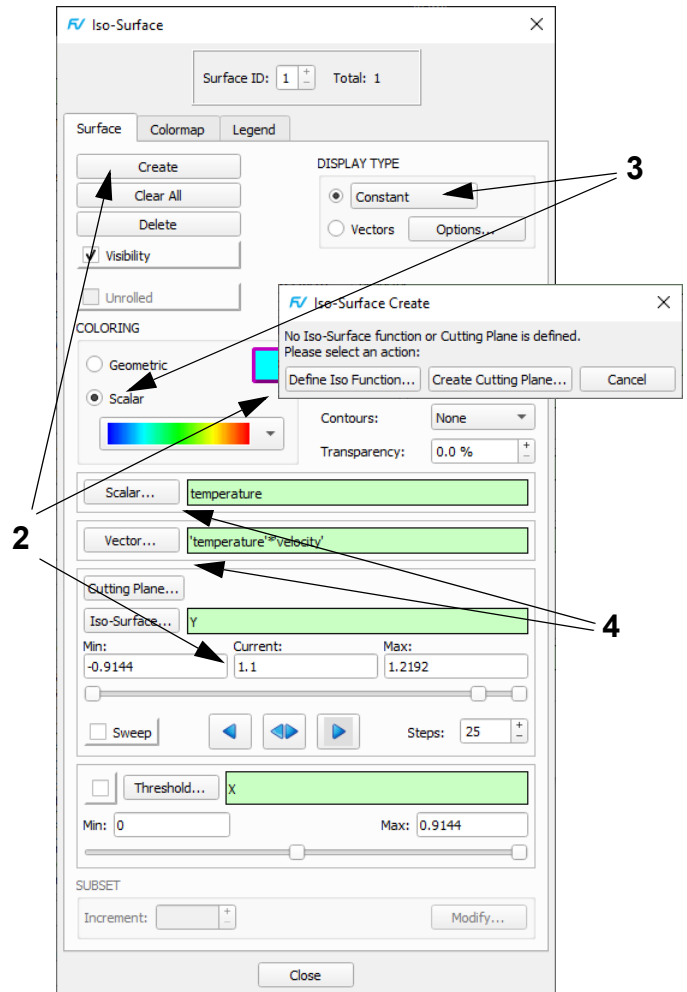
2,3

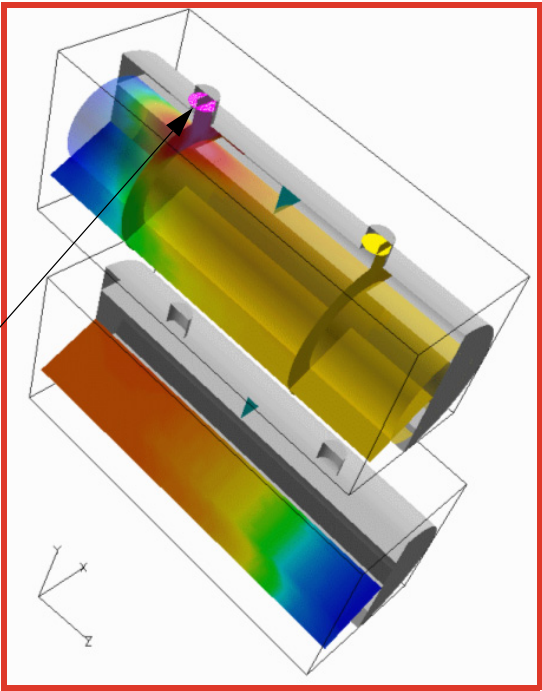
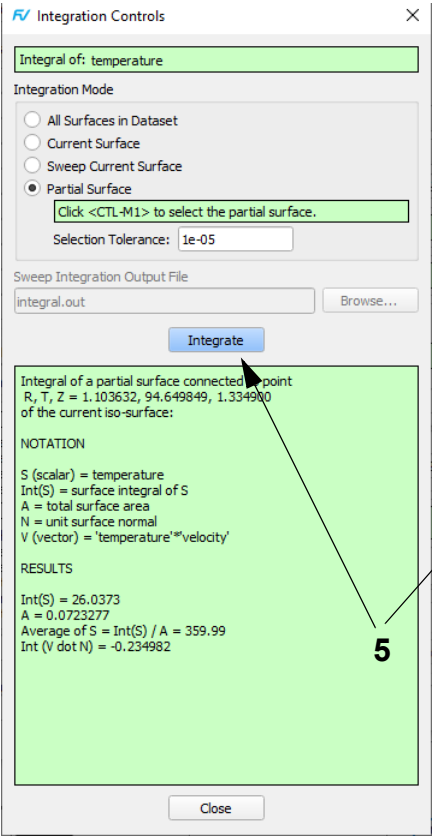


Step 4: Use Partial Integration for Convective Flow Calculations

In the following part of the tutorial we will show how to use the Partial Integration feature.

1. Make sure that Dataset 1 is current (if necessary, change the number to the right of **Dataset...** on the main toolbar OR **quick pick** a surface in Dataset 1). On the Viewer Toolbar, make sure the current **Object:** is **World**.
2. Navigate to **Visualization Panels** → **Iso-Surface...** OR click the **Iso** icon on the main toolbar to bring up the **Iso-Surface** panel. Click **Create**, and choose your Iso Function on the **Iso Surface Create** panel with **Define Iso Function**. Enter 1.1 in the **Current:** field OR move the slider to adjust the **Current:** value to 1.1 on the Iso-Surface Panel. (Note that creating additional Iso Surfaces on this panel will copy the current surface, rather than presenting the **Iso Surface Create Panel**.)
3. Rotate the model to get a better view of the iso-surface that intersects two protruding parts on the top of the heat exchanger. Make sure that for the iso-surface the **DISPLAY TYPE** is **Constant** shading and **COLORING** is **Scalar**.
4. Set the current **Vector Function** by clicking **Vector...** then on the **Function Selection** panel, select '**temperature**velocity**' and click **Calculate**. Set the **Scalar Function** by clicking **Scalar...** On the **Function Selection** panel, select **temperature** and click **Calculate**.
5. Navigate to **Tools** → **Integration...** OR expand the side toolbar and click the **Integrate** icon to bring up the **Integration Controls** panel. Set the **Integration Mode** to **Partial Surface**. Select (Ctrl + left click) the part of the iso-surface near the inlet. The section of the iso-surface will be highlighted. On the **Integration Controls** panel, click **Integrate**. The partial integration **RESULTS** near the inlet are displayed on the bottom of the **Integration Controls** panel. The average inlet temperature (Average of S) is 359.99; the inlet convective heat flow (Int(V dot N)) is -0.234982.





6. Select (Ctrl + left click) the part of the iso-surface near the outlet. On the **Integration Controls** panel, click **Integrate**. The partial integration **RESULTS** near the outlet are displayed on the bottom of the **Integration Controls** panel. The average outlet temperature (Average of S) is 323.679; the outlet convective heat flow (Int(V dot N)) is 0.210961.

Dataset 1 (cocurrent)	Temperature	Convective Heat Flow
Inlet	359.99	-0.234982
Outlet	323.679	0.210961

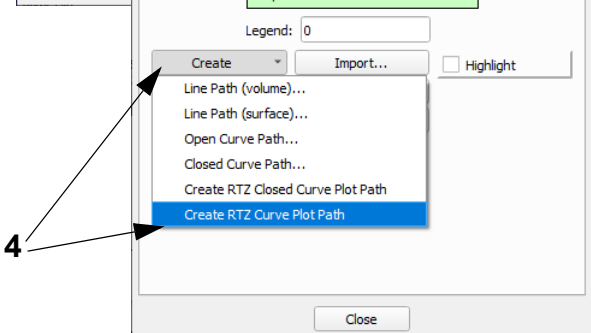
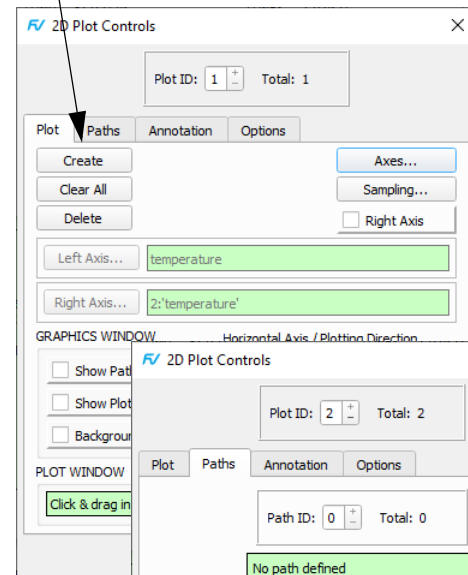
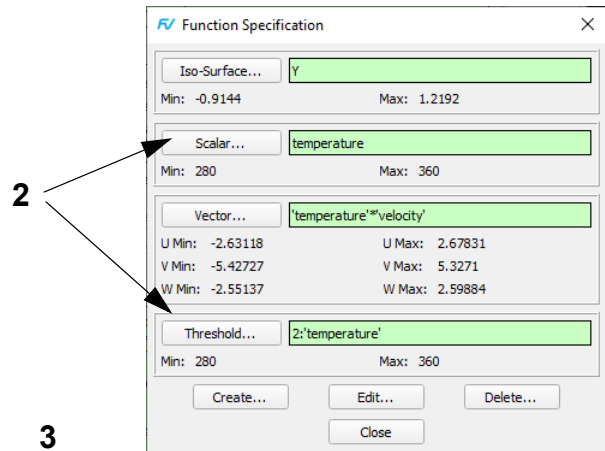
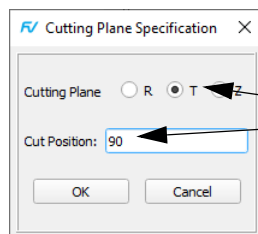
7. Repeat steps 2-6 for Dataset 2. The integration results for the second dataset are shown in the table. The results correspond to convective flux of heat in and out for two different heat exchanger configurations. The countercurrent configuration has a slightly lower value of convective flux at the outlet. These results suggest that there is a higher amount of heat exchange for this configuration, as expected.

Dataset 2 (countercurrent)	Temperature	Convective Heat Flow
Inlet	359.99	-0.234981
Outlet	311.372	0.202894

Step 5: Create a 2D Plot of Temperature

In the following part of the tutorial we will show how to create a simple 2D Plot of temperature along a straight line obtained as an intersection of two coordinate surfaces.

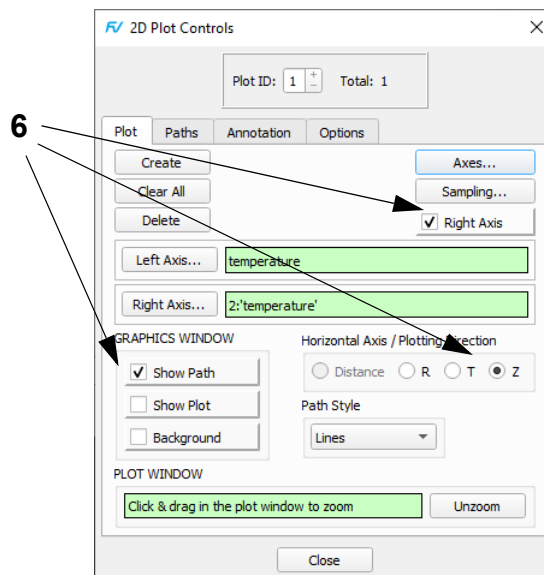
1. Make sure that Dataset 1 is current (if necessary, change the number to the right of **Dataset...** on the main toolbar OR **quick pick** a surface in Dataset 1).
2. Click the functions icon on the main toolbar to bring up the **Function Specification** panel. Make sure that **temperature** is selected for the **Scalar** register. Put **2:'temperature'** into the **Threshold** register.
3. Navigate to **Visualization Panels** → **2D Plots** OR expand the side toolbar and click the **Plot** icon to bring up the **2D Plot Controls** panel. On the **Plot** tab, click **Create** to create a 2D plot.
4. On the **Path** tab, click **Create** and select **Create RTZ Curve Plot Path** from the pulldown menu. As prompted by the **Cutting Plane Selection** panel, Ctrl + left click on the R=0.6 coordinate surface.
5. In the **Cutting Plane Specification** panel, select the **T** radio button, enter 90 for **Cut Position**, click **OK** and **Close**.

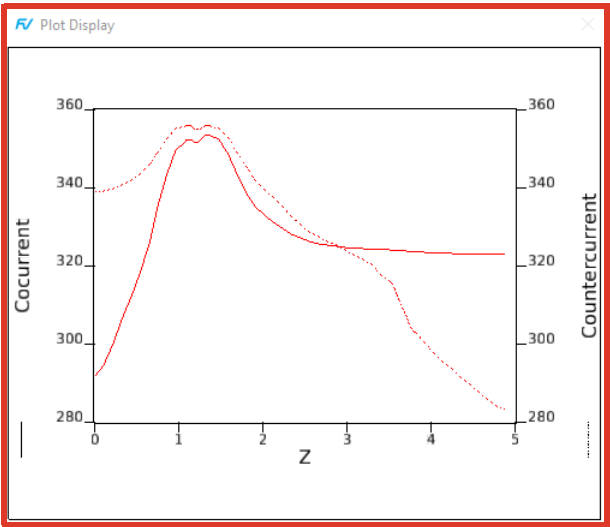
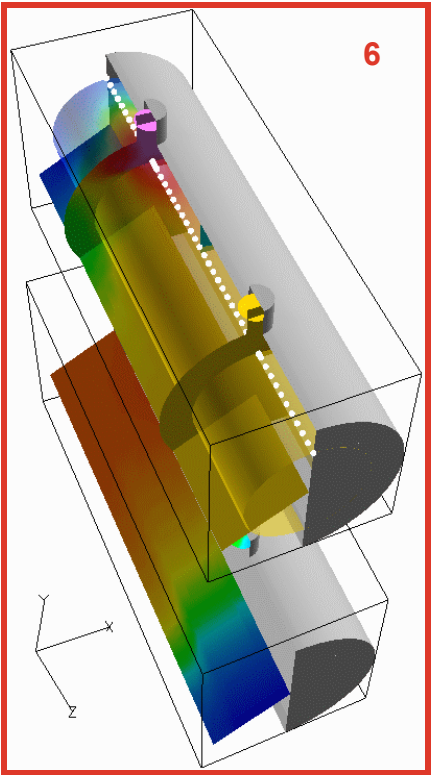


Step 5: Create a 2D Plot of Temperature (cont'd)

Here, we will finalize the 2D plot appearance.

6. On the **Plot** tab of the **2D Plot Controls** panel, turn on the **Right Axis** button. For **Right Axis Function**, click **Select...** In the **Function Selection** panel, select 2:'temperature' and click **Calculate**. Click **OK** to dismiss the **2D Plot Right Axis** warning. Turn on **Show Path** to see where the plot path is relative to the data-set. Change the **Horizontal Axis / Plotting Direction** to **Z** and click **OK** to dismiss the **2D Plot Horizontal Axis** warning.
7. Click the **Axes...** button. On the **Horiz Axis** tab, enter **TICK MARKS Major Unit: 1**. On the **Left Axis** tab, change **Label** to Cocurrent and enter **TICK MARKS Major Unit: 20**. On the **Right Axis** tab, change **Label** to Countercurrent and enter **TICK MARKS Major Unit: 20**. Click **Close**.
8. On the **Annotation** tab of the **2D Plot Controls** panel, turn off **Show Date**.



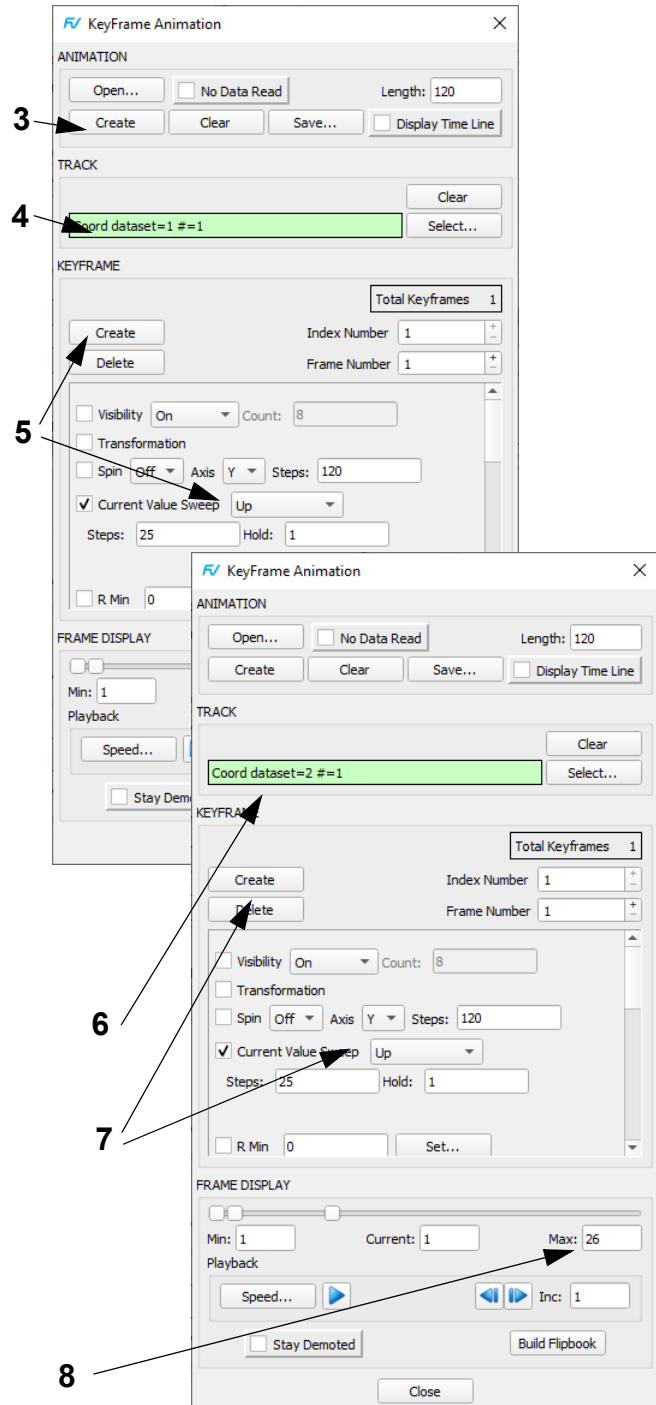


Step 6: Create a KeyFrame Animation using Multiple Datasets

Creating a KeyFrame animation using multiple datasets can be very helpful in visualizing your data particularly when comparing two designs.

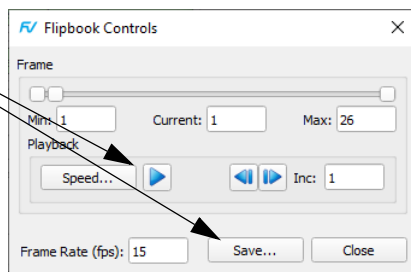
NOTE: This section of the tutorials offers good instruction on the use of the Keyframe interface for FieldView, however, sweeping Coordinate Surfaces across datasets no longer requires setting Keyframes. This can be more directly accomplished using **Linked Surface Sweep**, see [\[Page 307 Working with FieldView\]](#)

1. Make sure that Dataset 2 is current (if necessary, change the number to the right of **Dataset...** on the main toolbar OR **quick pick** a surface in Dataset 2). **Close** the **2D Plot Controls** panel (which will also close the **Plot Display**). Navigate to **File** → **Open Restart** → **Current Dataset...** to bring up the **OPEN RESTART: Current Dataset** browser. Select **start_on_one.dat** and click **Open** to re-read the Current Dataset Restart saved earlier. When the Current Dataset Restart is read, it automatically re-creates all surfaces previously stored with this restart on the second dataset.
2. Navigate to **Tools** → **KeyFrame Animation...** to bring up the **KeyFrame Animation** panel. [\[Page 316 Reference Manual\]](#)
3. In the **ANIMATION** area, click **Create** to create a new keyframe animation.
4. **Quick-pick** the theta coordinate surface in Dataset 1. The **Coordinate Surface** panel will come up behind the **Keyframe Animation** panel. Note that the **TRACK** field in the **KeyFrame Animation** panel has changed to **Coord dataset=1 #=1** (first surface of Dataset 1).
5. In the **KEYFRAME** area, click **Create** to create a keyframe for the current track. Change **Current Value Sweep** from **Off** to **Up**. This automatically puts a check mark next to **Current Value Sweep**.
6. **Quick-pick** the theta coordinate surface in Dataset 2. Note that the **TRACK** field in the **KeyFrame Animation** panel has changed to **Coord dataset=2 #=1** (first surface of Dataset 2).
7. In the **KEYFRAME** area, click **Create** to create a keyframe for the current track. Change **Current Value Sweep** from **Off** to **Up**. This automatically puts a check mark next to **Current Value Sweep**.
8. In the **FRAME DISPLAY** area, change **Max:** from 120 to 26; press **Enter** if typed in.

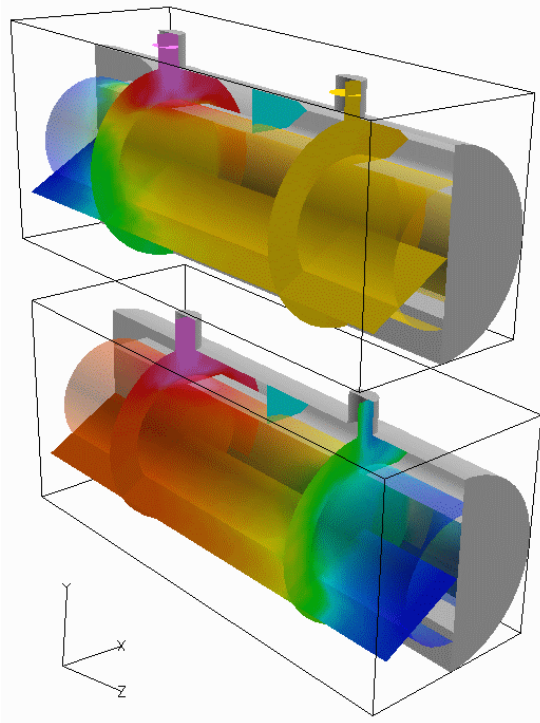


9. Click the Play toggle button to sweep both surfaces at the same time. Click the Play toggle button again to stop sweeping.
10. Optional: Navigate to **Tools** → **Graphics Layout Size** → **640x480 NTSC** to adjust the graphics window to the size accepted by most external movie players.
11. In the **KeyFrame Animation** panel's **FRAME DISPLAY** area, click **Build Flipbook**. A **Flipbook Display Size Warning** panel will come up; click **OK** to dismiss it. The **Building Flipbook** panel displays frame numbers as they are built and closes automatically when complete.
12. On the **Flipbook Controls** panel, click the Play toggle button to sweep both surfaces at the same time. Click the Play toggle button again to stop sweeping.
13. Click **Save...** to bring up the **Flipbook File Save** panel. Type in a **File name** and click **Save**. **Close** the **Flipbook Controls** panel.

12,13



12



Chapter 15

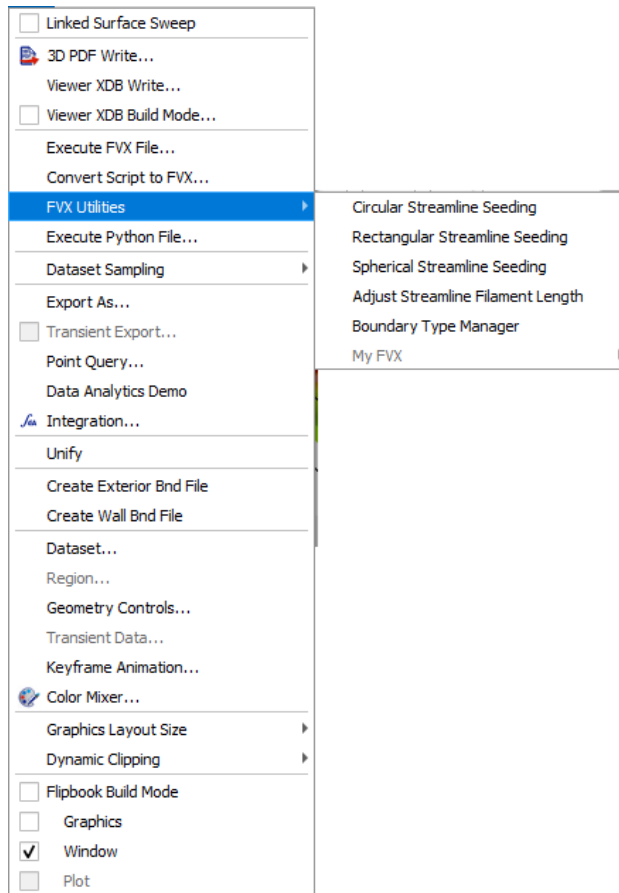
15

Streamline Seeding Tools

FieldView includes three streamline seeding tools that allow users to add spherical, rectangular and circular seeding geometries to datasets. The three Geometric Seed Distribution patterns utilize graphical user interfaces (GUIs) to provide quick, easy streamline seeding. Input options on the GUI panels determine the placement, orientation, size and seed quantity of the seeding geometries and allow the user to create new rakes as well as delete old rakes.

These **FVX** based tools are integrated into **FieldView** as part of the standard release. From the **Tools** menu, navigate to **FVX Utilities**. From the fly-out menu, you can choose one of the three seeding tools, as illustrated in the figure on the following page.

Detailed instructions on the use of spherical, rectangular and circular seeding are provided in the following sections. Multiple examples that utilize the sample tutorial datasets are also included for each geometry.



Spherical Seeding

This **FVX** script is a tool for inserting a sphere of streamline seeds into a pre-loaded dataset in **FieldView**. Running the script initiates an interactive GUI that provides placement and sizing options to the user. With this GUI, the user can choose the X, Y, and Z coordinates of the seeding sphere's center as well as its radius and bi-directional seed count. The GUI accepts these inputs, creating a rake of streamlines in the dataset.

How to Use Spherical Seeding

The simple process of running the spherical seeding **FVX** script and the subsequent GUI are outlined in the following steps. Examples of this process are provided in later sections.

1. Load the desired dataset into **FieldView** and visualize any boundary or coordinate surfaces needed to help reference the desired position of the seed sphere.
2. Select **Tools** → **FVX Utilities** → **Spherical Streamline Seeding**. This will launch the GUI for this streamline seeding tool.

3. The input fields on the **Spherical Seed Distribution** GUI are initially set to default values. A seed sphere created using these defaults will be positioned at the origin (center **X**, **Y**, and **Z** coordinates are all 0) with a **Radius** of 1 and **Phi** and **Theta Counts** of 4 and 12, respectively. The **Phi Count** represents the number of seeds placed longitudinally on one-quarter of the seeding sphere along a 180° arc. Similarly, the **Theta Count** represents the number of seed rings, each with twice the **Phi count** of seeds, positioned laterally “north” and “south” of the sphere's “equator” along a 90° arc. By default, a ring of 2n seeds is located at the sphere's equator, making the total number of seeds on the sphere the function of **Phi Count** “m” and **Theta Count** “n” shown in the expression below.

$$\text{Total Number of Seeds} = 4 \times m \times n + 2 \times n$$

Using default values (m = 4, n = 12) creates a sphere with 216 seeds. As a note, the overall seed spacing is most uniform when using an approximate m-to-n ratio of 2 to 5.

4. Make any desired changes to the GUI field values. For each change to register correctly with the interface, type the new value into the appropriate field and press **Enter**. Note that the **X**, **Y**, and **Z** components must describe a coordinate of the sphere's center that lies within the bounds of the dataset for the script to run (though it is acceptable to have seeds outside the dataset as long as the sphere's center remains within its bounds).
5. Press **Create Streamline Rake** to initiate seed placement and the streamline calculations.
6. The streamline rake along with the sphere of seeds should now appear in the **FieldView** graphics window at the appropriate location. Errors, if applicable, are printed in the terminal window from which **FieldView** was launched (Linux systems) or in the MS-DOS window (Windows systems).
7. The **Delete Streamline Rake** button on the GUI can be used at any time to delete the most recent streamline rake. Additional rakes with different characteristics can be created with the GUI by repeating Steps 4 and 5.

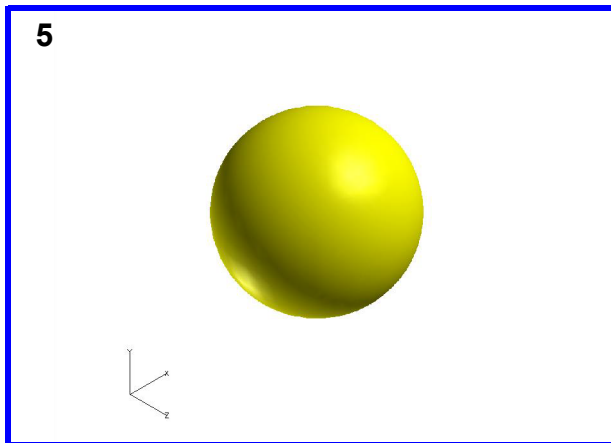
Spherical Seeding in the Feature Extraction Tutorial Dataset

The Feature Extraction Tutorial ([Chapter 13](#)) investigates the flow surrounding and within a water droplet. Because the water droplet is modeled here as a perfect sphere, the spherical seeding method is ideal for seeding streamline rakes inside and outside this geometry. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of **0.5**.

Visualizing the Boundary Surfaces

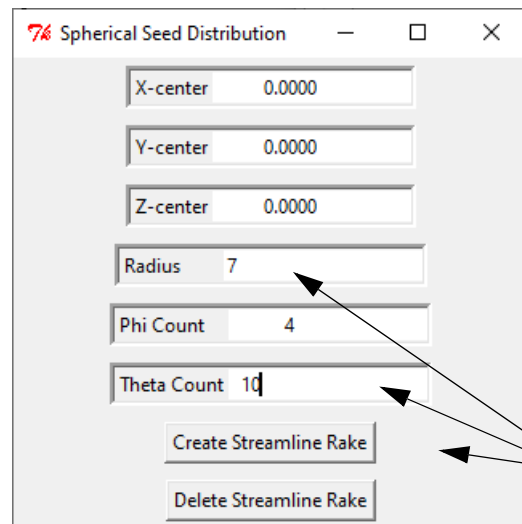
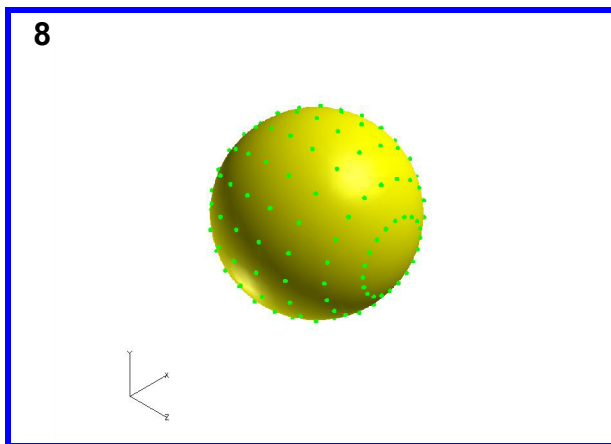


The pictures on each page correspond to the final image you should see when the steps are completed correctly.

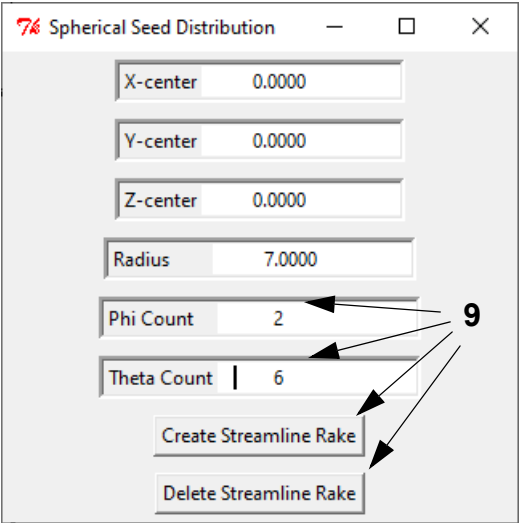
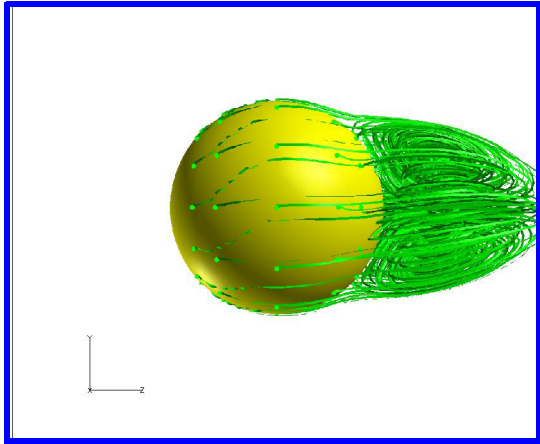


1. Bring up the **File** → **Data Input** → **AcuSolve [FV-UNS Export]** panel. Click **Read Grid or Combined Data**, select **recirc.uns** and click **Open**. Close the **AcuSolve [FV-UNS Export]** panel.
2. To visualize the geometry, click the side toolbar **Bound** icon OR use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. Select **BOUNDARY TYPES interface 1, interface 2 and interface 3** and click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
3. Change the **Geometric COLORING** to **Yellow**.
4. On the **Transform Controls** toolbar, click the icon to toggle **Outline** off.
5. On the **Transform Controls** toolbar, click the icon to **Center** the image.

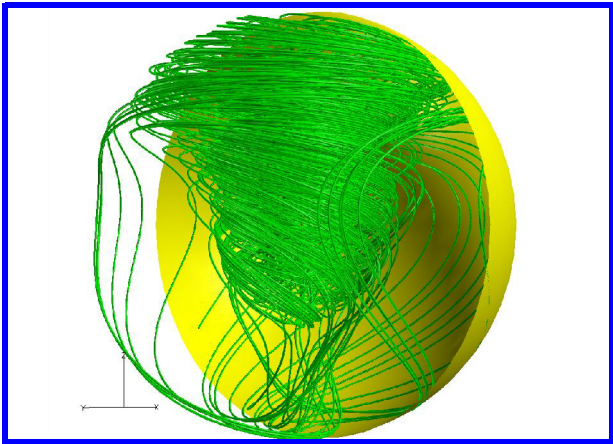
Exterior Flow Patterns



6. Bring up **Tools** → **FVX Utilities** → **Spherical Streamline Seeding** to run the script and initiate the GUI.
7. Since the water droplet is centered at the origin, keep the **X**, **Y**, and **Z** coordinates of the seeding sphere center as **0**. To best match that of the water droplet, change **Radius** to **7** and press **Enter**. Change **Theta Count** to **10** and press **Enter**. After making the two changes, click **Create Streamline Rake**.
8. In a few seconds, the seeding sphere and streamline rake should appear. On the **Streamlines** panel, turn the **Visibility** off to view the seed sphere.
9. On the **Spherical Seed Distribution** GUI, click **Delete Streamline Rake**. Change **Phi Count** to **2** and press **Enter**; change **Theta Count** to **6** and press **Enter**. Click **Create Streamline Rake** to produce a less cluttered visualization of the exterior flow.
10. On the **Streamlines** panel, change **DISPLAY TYPE** from **Complete** to **Ribbons**. Use the **View** menu OR click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Click **-X** and **Close** the panel. Change the **Geometric COLORING** to **Green**. Observe the vortex core ring just beyond the aft end of the water droplet.

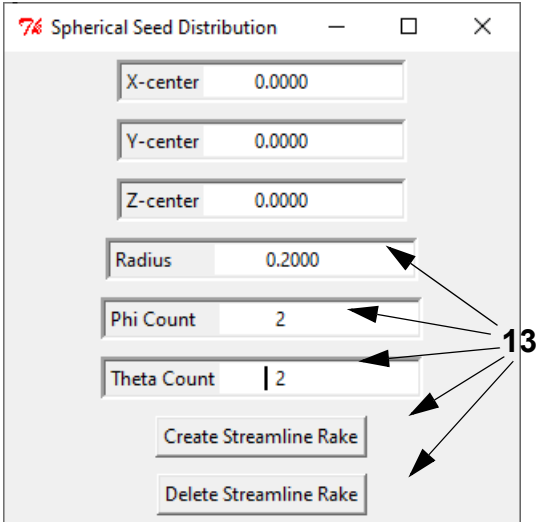


Interior Flow Patterns



11. Click **Dataset...** on the main toolbar OR navigate to **Tools → Dataset...** On the **Dataset Controls** panel, set **X ROTATE** to **-90°**, and **Close**.
12. “Quick-pick” (double-click) the water droplet boundary surface. Click the main toolbar **f_x** icon. On the **Function Specification** panel, click **Threshold...** On the **Function Selection** panel, select **X** and click **Calculate**. **Close** the **Function Specification** panel. On the **Boundary Surface** panel, change **Min:** to **0.0** and turn **Threshold Clip** on. As in the Feature Extraction Tutorial (**Chapter 13**), this allows the user to view the interior of the water droplet by removing half of the droplet’s boundary.

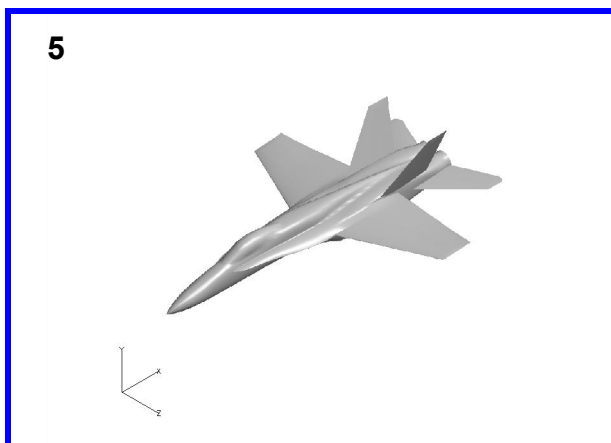
13. On the **Spherical Seed Distribution** GUI, click **Delete Streamline Rake**. To seed a spherical rake on the inside of the droplet, reduce the **Radius** to **0.2** and both the **Phi Count** and **Theta Count** to **2**, pressing **Enter** after changing each value. Click **Create Streamline Rake**.
14. On the **Streamlines** panel, in the **Calculation Parameters** section, change the **Direction** to **Both**. Click **Calculate** to readjust the streamlines. With the mouse controls (**M2** and **M3**), rotate and zoom the droplet to the desired view.



Spherical Seeding in the Basic Aerospace Tutorial Dataset

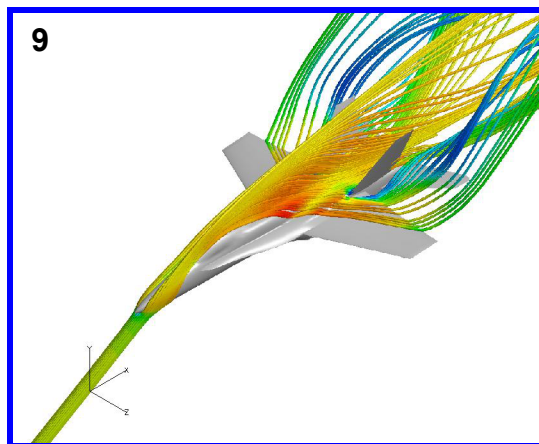
The Basic Aerospace Tutorial ([Chapter 3](#)) investigates the flow field surrounding an F18 at a high angle of attack in subsonic flight. Applying the spherical seeding method to this dataset allows regions of interest to be analyzed where conventional seeding techniques would be difficult. The following steps detail the use of spherical seeding to investigate flow around the nose of the aircraft, the 3D aerodynamic effects visible at the wing tips and the interference effects of the wing-fuselage junction. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of **0.5**. The F18 data is courtesy of NASA Langley Research Center. Only applies to **FieldView**.

Visualizing the Boundary Surfaces



1. Bring up the **File** → **Data Input** → **PLOT3D** panel. Check both **IBlanks** and **Multi-Grid**. Click **Read XYZ Data...** In the file browser, select **f18i9b_g_bin** and click **Open**. On the **Grid Subset Selection** panel, click **OK** to select all 9 grids. Click **Read Q Data...**
2. Select **f18i9b_q_bin** and click **Open** to read in the results data. **Close** the **PLOT3D** panel.
3. Click the side toolbar **Bound** icon OR use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. Select **BOUNDARY TYPES** **body**, **wing** and **tail** and click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
4. On the **Transform Controls** toolbar, click **Viewer Options...** Turn **Outline** and **Perspective** off. **Close** the **Viewer Options** panel.
5. Click **Dataset...** on the main toolbar OR navigate to **Tools** → **Dataset...** to bring up the **Dataset Controls** panel. Set **DUPLICATION** to **Mirror**. On the **Dataset Mirror Parameters** panel, turn **X** off and turn **Z** on. Click **OK** then **Close**. **Close** the **Dataset Controls** panel.
5. The full aircraft should now appear in the graphics window. From the **Transform Controls** toolbar, click the **Center** icon to center and zoom in on the F18.

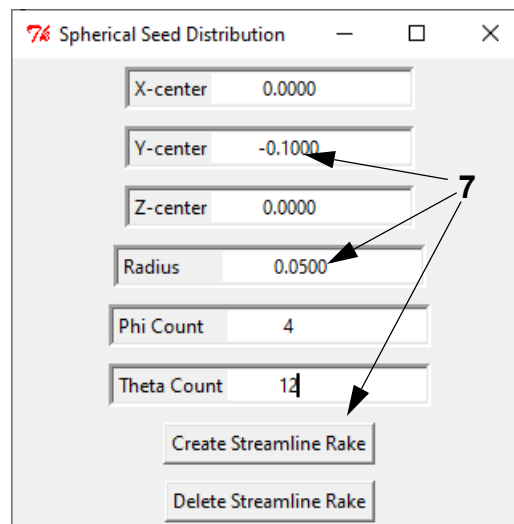
Nose Tip Flow Patterns



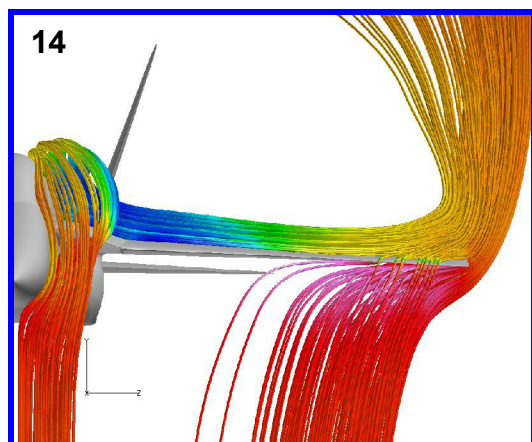
6. Select **Tools** → **FVX Utilities** → **Spherical Streamline Seeding** to run the script and initiate the GUI.
7. The nose-tip of the aircraft is located at the XYZ location (0, 0, 0). Since the aircraft is at a high angle of attack, the region just above the nose tip in the Y-direction is of little importance, as streamlines passing through this location will be affected minimally. For this reason, the seed sphere will be located slightly below the tip of the nose. On the Spherical Seed Distribution GUI, change **Y-center** to **-0.1** and press **Enter**. Change **Radius** to **0.05**, **Phi** and **Theta** of **4** and **12**, pressing **Enter** for each. Click **Create Streamline Rake**.
8. On the **Streamlines** panel, in the **Calculation Parameters** section, change the **Direction** to **Both**. Click

Calculate to readjust the streamlines. Turn **Display Seeds** off to hide the seed sphere.

9. To view the streamlines as a function of Mach number, click the main toolbar **f_x** icon. Click **Scalar...** On the **Function Selection** panel, select **Mach Number [PLOT3D]** and click **Calculate**. Note that the F18's flow field has a Mach number range of approximately 0 to 0.6. **Close** the **Function Specification** panel. On the **Streamlines** panel, change the **COLORING** from **Geometric** to **Scalar**. To adjust the color scheme, click the **Colormap** tab (OR the main toolbar icon OR **Edit** menu to bring up the **Scalar Colormap Specification** panel). Change the **Colormap** to **NASA-2** and confirm that the minimum and maximum spectrum values reflect the approximate minimum and maximum flow field Mach numbers of 0 and 0.6, respectively.

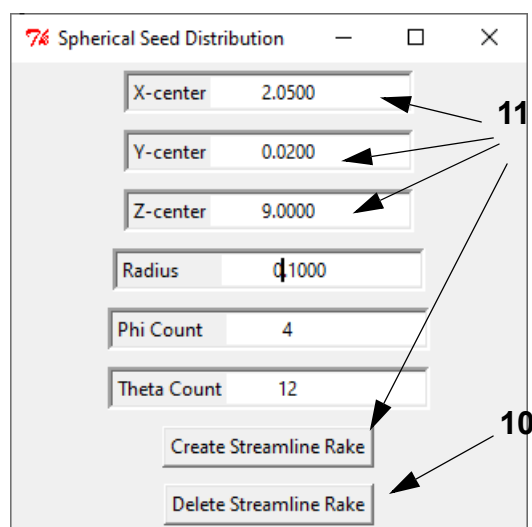


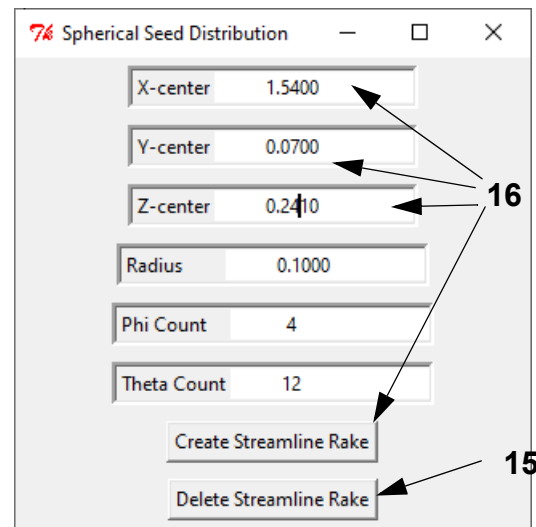
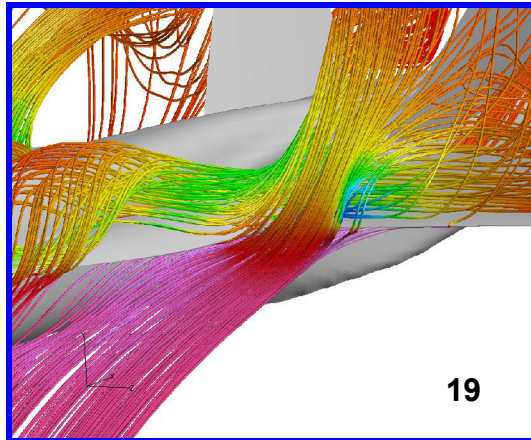
3-D Aerodynamic Effects at the Wing Tips



top of the wing and the high pressure streamlines (pink) below the wing both converge to an average pressure value shown in orange at the wing tip. This is to say that the pressure differential between the high pressure below and the low pressure above the wing that gives rise to aerodynamic lift is reduced as the spanwise position approaches the wing tip.

10. On the **Spherical Seed Distribution** GUI, click **Delete Streamline Rake** to clear the previous streamline rake.
11. The center for the new seed sphere is located at mid-chord on the left wing, close to the wing tip. To set the center, change **X-center** to **2.05**, **Y-center** to **0.03** and **Z-center** to **0.9**. Adjust the **Radius** to **0.1**, remembering to press **Enter** after each change. Click **Create Streamline Rake**.
12. On the **Streamlines** panel **Rake** tab, change the **Direction** to **Both** and click **Calculate**. Turn **Show Seeds** off to hide the seed spheres. Note that because the dataset is reflected, a rake of streamlines appears on each wing.
13. To view the streamlines as a function of pressure coefficient, click the main toolbar **f_x** icon. Click **Scalar...** On the **Function Specification** panel, select **Cp[PLOT3D]** and click **Calculate**. **Close** the **Function Specification** panel. On the **Streamlines** panel, change the **COLORING** to **Scalar** if necessary. To adjust the color scheme, click the **Colormap** tab (OR the main toolbar icon OR Edit menu to bring up the **Scalar Colormap Specification** panel). The **Colormap** should be set to **NASA-2**. The maximum spectrum value should reflect that of the maximum pressure coefficient in the flow field (about 0.7); adjust the minimum spectrum value to **-1.85** to obtain a better contour distribution.
14. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Set the **VIEWING DIRECTION** to **-X** and **Close** the panel. Using the mouse controls (**M1** and **M3**), translate and zoom to view the leading edge of the left wing. Locating the seed sphere at its current position allows streamlines to form on the top and bottom surfaces of the wing. Note that the low pressure streamlines (blue and green) on



Interference Effects at the Wing-Fuselage Junction

15. On the **Spherical Seed Distribution** GUI, click **Delete Streamline Rake** to clear the previous streamline rake. On the **Transform Controls** toolbar, click the icon to re-center the aircraft.
16. The center for the new seed sphere is located at the leading edge of the left wing's root chord. To set the center, change **X-center** to **1.54**, **Y-center** to **0.07** and **Z-center** to **0.241**, pressing **Enter** after inputting each value. Click **Create Streamline Rake**.
17. On the **Streamlines** panel **Rake** tab, change the **Direction** to **Both** and click **Calculate**. Turn **Show Seeds** off to hide the seed spheres.
18. The **Scalar Function** should still be **Cp [PLOT3D]**. Set the **COLORING** to **Scalar**. On the **Colormap** tab, move the bottom slider all the way down to readjust the minimum spectrum value to the range limit of **-3.53526**.
19. Using the mouse controls (**M2** and **M3**), rotate and zoom to view the section of the fuselage that joins the left wing. Like in the previous example, locating the seed sphere at its current position provides streamlines on both the top and bottom surfaces of the wing-fuselage junction. The high pressure streamlines (pink and red) that hug the underside of the fuselage ahead of the wing see a sudden drop in pressure at the junction, having to accelerate around the geometry of the wing root. This forced acceleration of the flow indicates an expected drag rise at this location on the F18.

Rectangular Seeding

This **FVX** script is a tool for inserting a rectangular panel of streamline seeds into a pre-loaded dataset in **FieldView**. Running the script initiates an interactive GUI that provides placement and sizing options to the user. With this GUI, the user can choose the X, Y and Z coordinates of the seeding rectangle's center, the direction normal to the panel (X, Y or Z), the dimensions of the panel and its bi-directional seed count. The GUI accepts these inputs, creating a rake of streamlines in the dataset.

How to Use Rectangular Seeding

The simple process of running the rectangular seeding **FVX** script and the subsequent GUI are outlined in the following steps. Examples of this process are provided in later sections.

1. Load the desired dataset into **FieldView** and visualize any boundary or coordinate surfaces needed to help reference the desired position of the seed rectangle.
2. Select **Tools** → **FVX Utilities** → **Rectangular Streamline Seeding**. This will launch the GUI for this streamline seeding tool.
3. The eight input fields on the **Rectangular Seed Distribution** GUI are initially set to default values. A rectangular seed panel created using these defaults will be positioned at the origin (center **X**, **Y** and **Z** coordinates are all 0) perpendicular to the **X** direction having **Width** and **Height** dimensions of 1 with 12 seeds positioned along each direction. Choosing the direction normal to the panel as one of the axes fixes which axis corresponds to the width and which axis corresponds to the height as shown in the table below.

Normal	Width	Height
X	Y	Z
Y	X	Z
Z	X	Y

A range of values (minimum and maximum) can be entered into the **Width** and **Height** fields to fix the locations of the width and height endpoints in the dataset. If a range is input for a given direction (**Height** or **Width** as X, Y or Z), the corresponding direction's "center" point value (**X-center**, **Y-center** or **Z-center**) becomes irrelevant as the values input as the range fully define the dimension. The seeds placed along the width and height of the seeding rectangle (as indicated by the **Width** and **Height Counts**) are evenly spaced along their respective sides. If "m" seeds are seeded along the width and "n" seeds are seeded along the height, the total number of seeds produced in the rake is the product of "m" and "n", provided the rectangle's center, width and height settings generate a seeding panel that lies within the dataset (although this is not required for the script to run).

$$\text{Total Number of Seeds} = m \times n$$

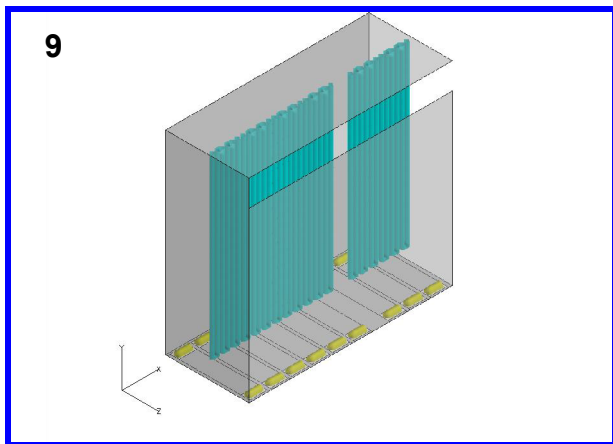
Using default values ($m = 12$, $n = 12$), one would create a rectangular panel with 144 seeds.

4. Make any desired changes to the GUI field values. Note that the three possible inputs to the **Set Normal** field are case sensitive (capital letters only). For each change to register correctly with the interface, type the new value into the appropriate field and press **Enter**. As mentioned above, it is acceptable to have the center of the rectangle or any seeds outside the dataset. When making use of this property, note that seeds lying outside the boundaries of the dataset will not be included in the rake that is produced.
5. Press **Create Streamline Rake** to initiate seed placement and the streamline calculations.
6. The streamline rake along with the panel of seeds should now appear in the **FieldView** graphics window at the appropriate location. Errors, if applicable, are printed in the terminal window from which **FieldView** was launched (Linux systems) or in the MS-DOS window (Windows systems).
7. The **Delete Streamline Rake** button on the GUI can be used at any time to delete the most recent streamline rake. Additional rakes with different characteristics can be created with the GUI by repeating Steps 4 and 5.

Rectangular Seeding in the Basic Combustion Tutorial Dataset

The Basic Combustion Tutorial ([Chapter 6](#)) investigates the flow within a propane combustion chamber. Because the chamber is a rectangular prism itself, the rectangular seeding method is ideal for seeding streamline rakes at various points within this geometry. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of **0.5**. Only applies to **FieldView**.

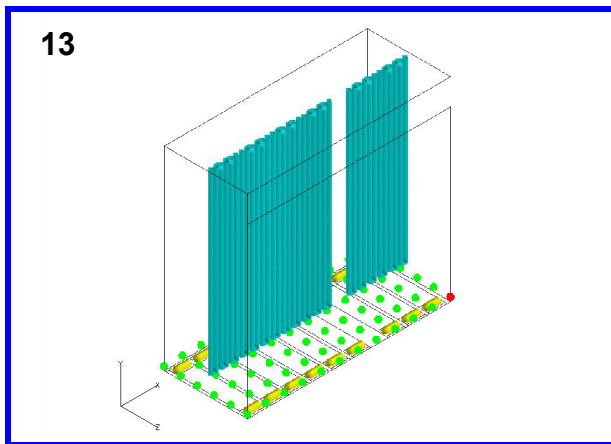
Visualizing the Boundary Surfaces, Pipes, and Burners



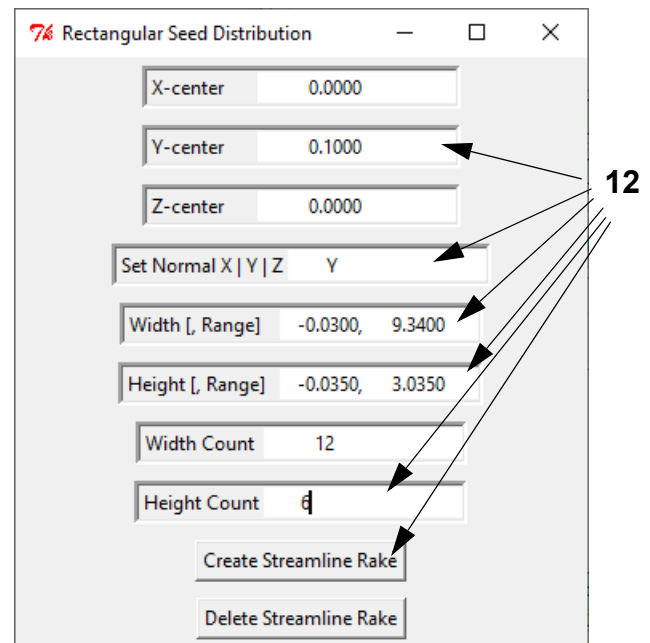
1. Bring up the **File** → **Data Input** → **PLOT3D** panel. Click **Read XYZ Data...** In the file browser, select **sw_xyz** and click **Open**. On the **Grid Subset Selection** panel, click **OK** to select the grid. Click **Read Function Data...** Select **sw_fun** and click **Open**. On the **Function Name Input** panel, click **Open Function Name File...** Select **sw.nam** and click **Open**. Close the **PLOT3D** panel.
2. Click the side toolbar Bound icon OR use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. Select **BOUNDARY TYPE wall** and click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading. Change the **TRANSPARENCY** to **50%**.
3. **Create** a second **wall** boundary surface. Change the **DISPLAY TYPE** to **Outlines**.
4. On the **Transform Controls** toolbar, click **Viewer Options...** Turn **Outline** and **Perspective** off. Close the **Viewer Options** panel. Change the **Geometric COLORING** to **Black**.
5. On the main toolbar, click **Dataset...** to bring up the **Dataset Controls** panel. Change **X SCALE** to **1.2** and **Z SCALE** to **1.5**. and **Close** the panel. Use the mouse

- controls **M1** and **M3** to translate and zoom the view as needed.
6. Click the side toolbar **Iso** icon OR use the **Visualization Panels** menu to bring up the **Iso-Surface** panel. Click **Create**. When prompted, click **Define Iso Function...**, select **walls** and click **Calculate**. Change the **Iso Function Current** value to **6** and press **Enter**. Ensure the **SUBSET Increment** is **1** and press **Enter** if necessary. Change the **DISPLAY TYPE** from **Constant** to **Smooth** shading.
 7. Change the **Geometric COLORING** to **Yellow**. These are the **burner inlets**.
 8. **Create** a second iso-surface and change the **Iso Function Current** value to **7**. These are the **tube coils** and **external walls**. Change the **Geometric COLORING** to **Light Blue**.
 9. In the **SUBSET** section, click **Modify...** to bring up the **Iso-Surface Subset Parameters** panel. Change all three **Min:** values to **2** and decrease each **Max:** value by one, to **113**, **27** and **20**, respectively. Click **OK** to visualize the tube coils only and **Close** the panel.

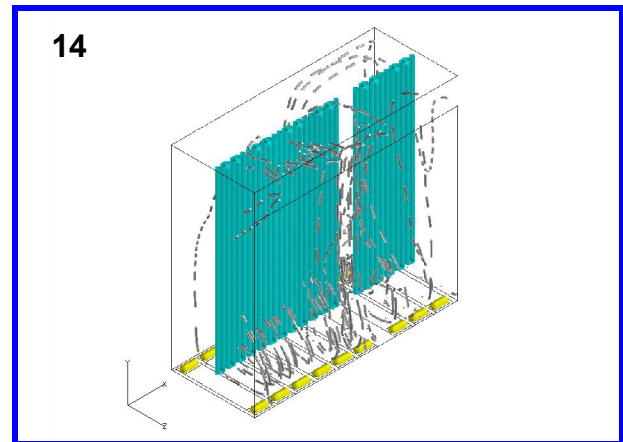
Flow Patterns Around the Burner Inlets



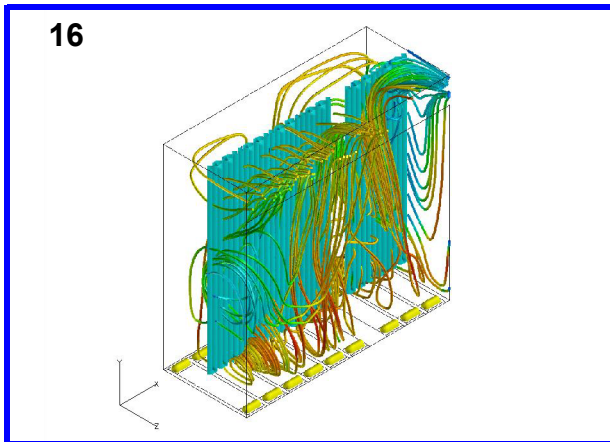
10. "Quick-pick" (double-click) the outside wall boundary to bring up the **Boundary Surface** panel for Surface 1. Turn the **Visibility** off.
11. Bring up **Tools** → **FVX Utilities** → **Rectangular Streamline Seeding** to run the script and initiate the GUI.
12. Change **Y-center** to **0.1** and press **Enter**. Change **Set Normal** to **Y** and press **Enter**. This will position the rectangle perpendicular to the Y-axis just above the burner tops. To create a seeding panel that extends to the four walls of the chamber, the minimum and maximum width (X) and height (Z) values must correspond to the minimum and maximum bounds of the dataset in these respective directions as follows: Change **Width [, Range]** to **-0.03, 9.34** and press **Enter**. Change **Height [, Range]** to **-0.035, 3.035** and press **Enter**. Change **Height Count** to **6** and press **Enter** for more even seed spacing. Click **Create Streamline Rake**.
13. In a few seconds, the seeding rectangle and streamline rake should appear. On the **Streamlines** panel, turn the **Visibility** off to view the seeding panel.



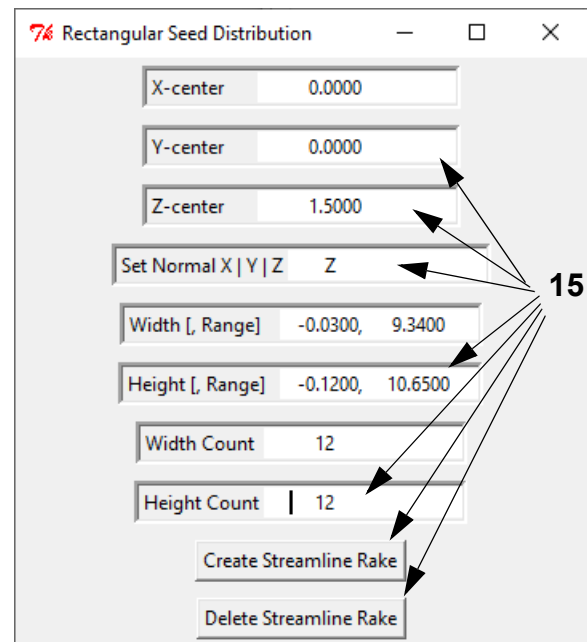
14. Turn the streamline **Visibility** back on and turn **Show Seeds** off. Change the **DISPLAY TYPE** from **Complete** to **Filament** and increase the animation divisions **Div:** to **200**. Turn **Animate** on to view the flow pattern from the burners.



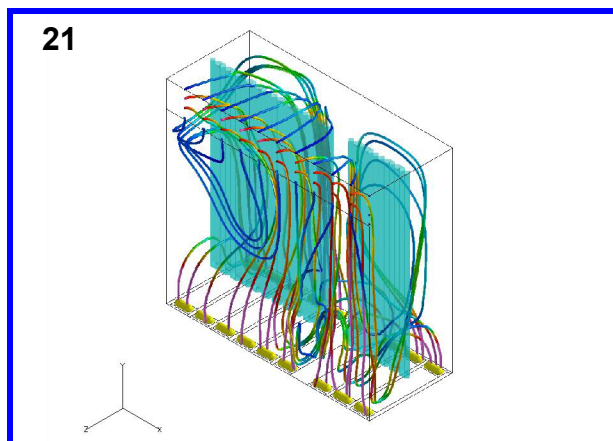
Flow Patterns Around the Pipes



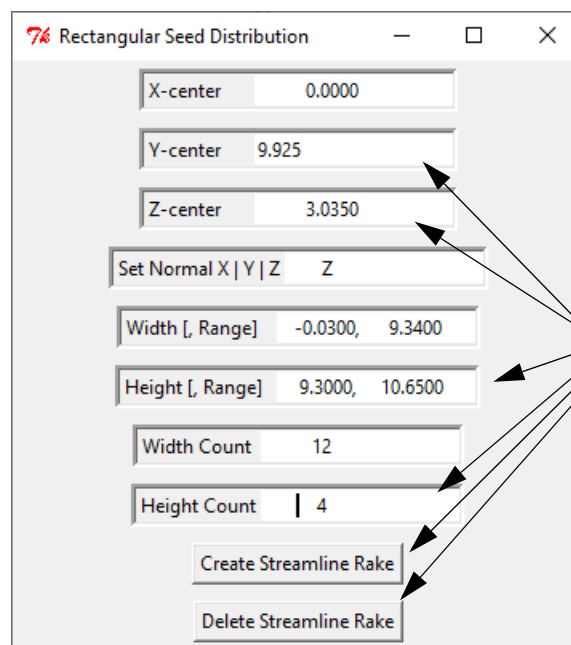
15. On the **Rectangular Seed Distribution** GUI, click **Delete Streamline Rake**. To seed a rectangular rake along the pipes, reset **Y-center** to **0**, change **Z-center** to **1.5** and **Set Normal** to **Z**. Change **Height [, Range]** (now representing the Y-axis) to **-0.12, 10.65** and reset **Height Count** to **12**, remembering to press **Enter** after each change. Click **Create Streamline Rake**.
16. Click the main toolbar **f_x** icon. On the **Function Specification** panel, click **Scalar...** Select the function **gas_temp** and click **Calculate**. Close the **Function Specification** panel. On the **Streamlines** panel, change **COLORING** to **Scalar** and turn **Show Seeds** off. On the **Colormap** tab, change the **Colormap** from **Spectrum** to **NASA-1**. Adjust the minimum spectrum value to **1200** and the maximum to **1800**.



Combustion Chamber Exit Flow



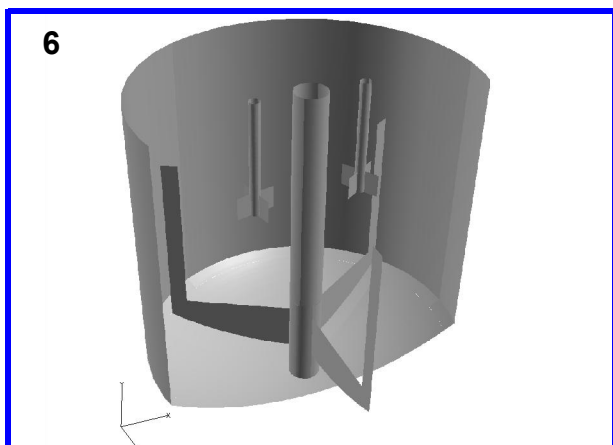
17. On the **Rectangular Seed Distribution** GUI, click **Delete Streamline Rake**. To seed a rectangular rake at the exit of the combustion chamber, change **Y-center** to **9.925** and **Z-center** to **3.035**. Change **Height [, Range]** to **9.3, 10.65** to reflect the extents of the exit duct in the Y-direction. Reduce **Height Count** to **4**, remembering to press **Enter** after each change. Click **Create Streamline Rake**. (Note that changing **Y-center** to any value and seeding streamlines will create exactly the same rake as the current rake. By specifying a **Height [, Range]** in the Y direction, specifying a **Y-center** point from which **Height** can be dimensioned is unnecessary).
18. Initially, no streamlines appear because the current seeding panel is placed at the exit of the combustion chamber and the default setting for the streamline **Direction** is **Forward**. On the **Streamlines** panel **Rake** tab, change the **Direction** to **Backward** to trace the streamlines backwards from the seeding locations. Click **Calculate** to visualize the streamlines.
19. Turn **Show Seeds** off and change the **COLORING** to **Scalar**. For **Scalar Function**, click **Select...** On the **Function Selection** panel, select **vel_mag** and click **Calculate**. On the **Colormap** tab, change the maximum to **6**.
20. "Quick-pick" (double-click) the pipes (iso-surface 2) and set the **Transparency** to **50%**.
21. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Set the **ISOMETRIC** view to **+X +Y +Z** and **Close** the panel.



Rectangular Seeding in the Chemical Processing Tutorial Dataset

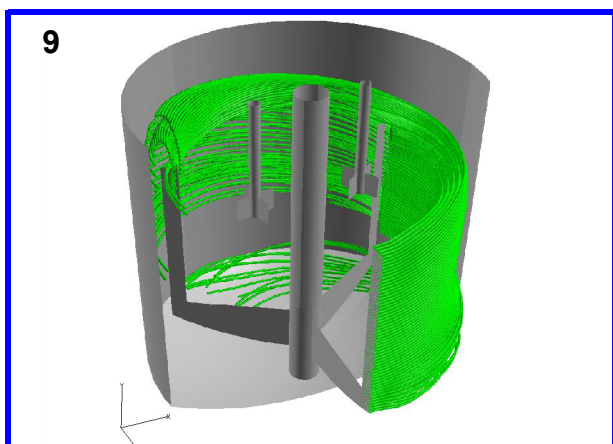
The Chemical Processing Tutorial ([Chapter 11](#)) investigates the flow field within a three-blade mixer. Applying the rectangular seeding utility allows the user to efficiently seed the flat, rectangular surfaces of the mixer blades. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of 0.5.

Visualizing the Boundary Surfaces

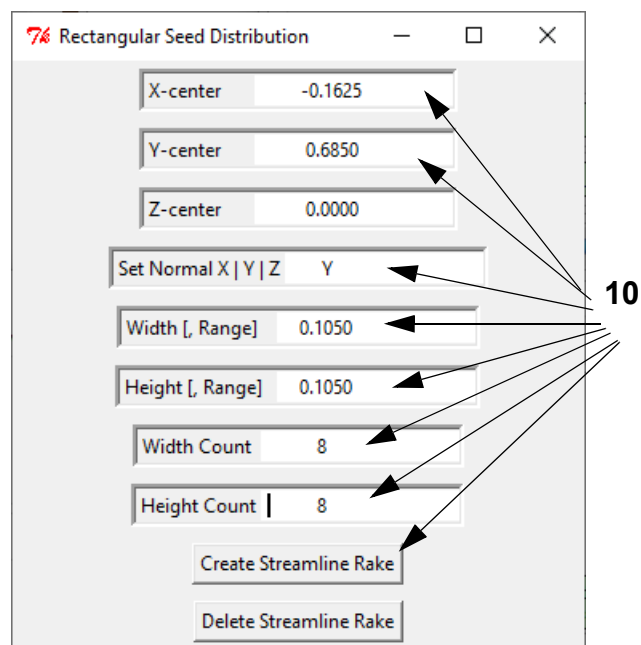
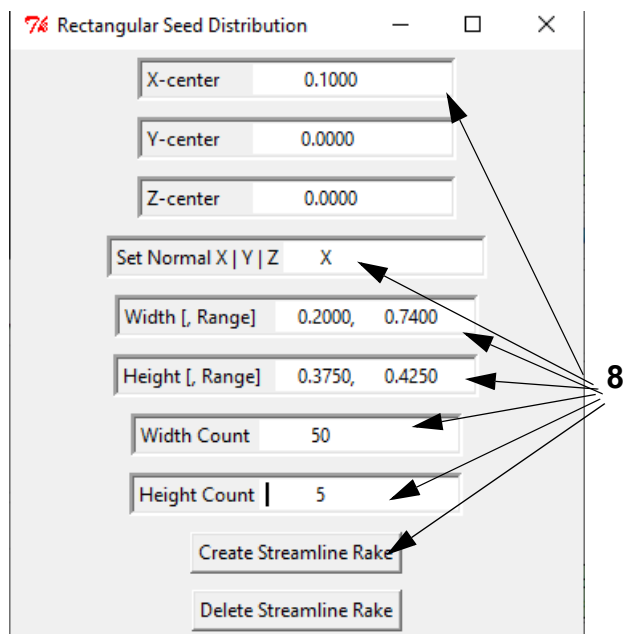


1. Bring up **File** → **Data Input** → **More Readers** → **CFX-4[dmp]**. Click **Read Grid & Results Data...** In the file browser, select **anchor-mixer_dmp** and click **Open**. On the **Function Subset Selection** panel, click **OK** to select all functions. Close the **CFX-4[dmp]** panel.
2. Click the side toolbar **Bound** icon to bring up the **Boundary Surface** panel and click **Create**. Hold down the Ctrl key to select **name=WALL-ANCHOR** and **name=WALL-SHAFT**. Click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
3. On the **Transform Controls** toolbar, click the icon to toggle **Outline** off.
4. **Create** a second boundary surface to include **name=WALL-BLADE1**, **name=WALL-SMALL-SHAFT1**, **name=WALL-BLADE2** and **name=WALL-SMALL-SHAFT2**. Click **OK**.
5. **Create** a third boundary surface with **name=WALL**. For **Threshold Function**, click **Select...** On the **Function Selection** panel, select **Z** and click **Calculate**. Turn **Threshold Clip** on and change **Max:** to **0.125**.
6. Use the mouse controls **M2** and **M3** to produce the view shown.

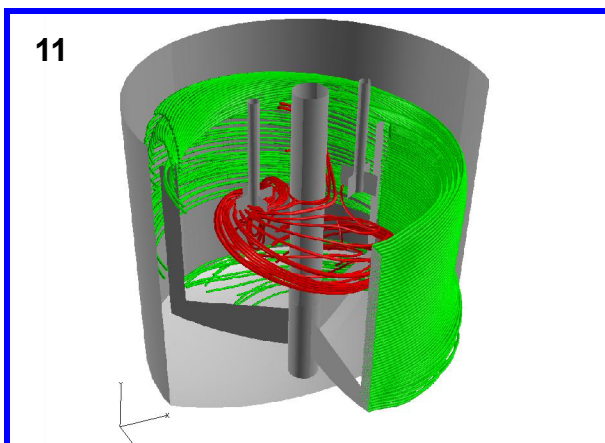
Visualizing the Mixer Flow Field

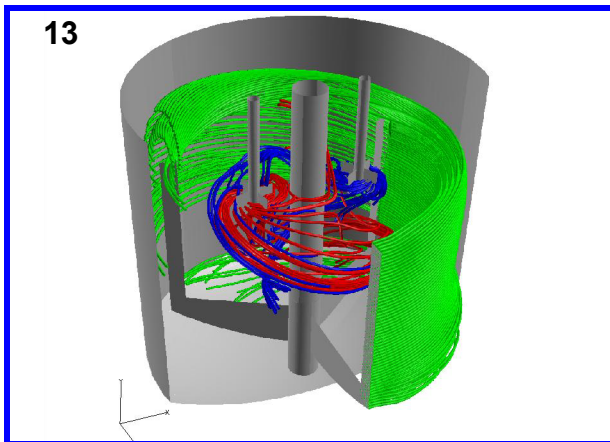


7. Bring up **Tools** → **FVX Utilities** → **Rectangular Streamline Seeding** to run the script and initiate the GUI.
8. The anchor blade closest in the current view is oriented exactly perpendicular to the X-axis. To seed a rake of streamlines on this blade, input the following on the **Rectangular Seed Distribution** GUI: Change **X-center** to **0.01**, press **Enter** and ensure **Set Normal** is **X**. Change **Width [, Range]** to **0.2, 0.74** and press **Enter**. Change **Height [, Range]** to **0.375, 0.425** and press **Enter**. Change **Width Count** to **50** (and press **Enter**) and **Height Count** to **5** (and press **Enter**) for even seed spacing. Click **Create Streamline Rake**.
9. On the **Streamlines** panel, in the **Calculation Parameters** section, turn **Time Limit** on, set it to **8** and press **Enter**. Click **Calculate**. Turn **Show Seeds** off to hide the seed rectangle. Change the **Geometric COLORING** to **Green**.

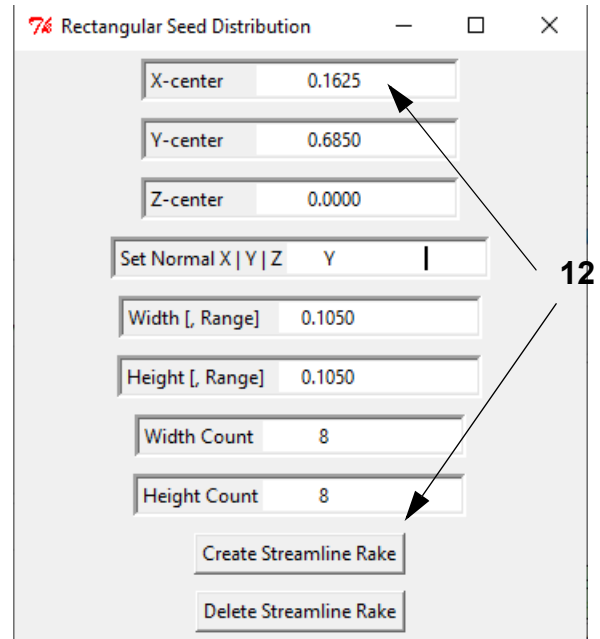


10. On the **Rectangular Seed Distribution** GUI, adjust the settings to create a rectangular streamline seed panel at the left small mixer blade as follows: Change **X-center** to **-0.1625**, **Y-center** to **0.685** and **Set Normal** to **Y**. Change **Width** and **Height** to **0.105** (use magnitudes instead of ranges). Change **Width Count** and **Height Count** to **8**, pressing **Enter** after each change. Click **Create Streamline Rake**.
11. On the **Streamlines** panel, turn **Time Limit** on, set it to **8** and press **Enter**. Click **Calculate**. Turn **Show Seeds** off and change the **Geometric COLORING** to **Red**.
12. On the **Rectangular Seed Distribution** GUI, change **X-center** to **0.1625** and press **Enter** to create an identical rectangular streamline seed panel at the right small mixer blade. Click **Create Streamline Rake**.

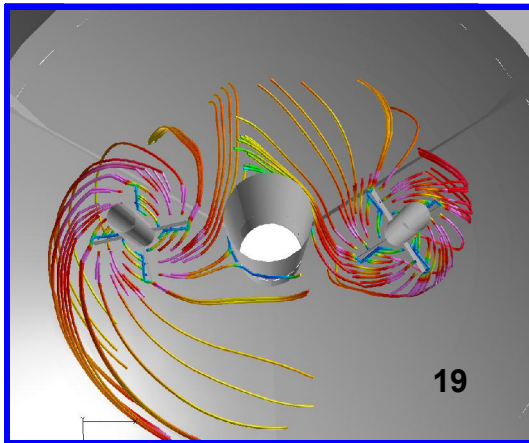




13. On the **Streamlines** panel, turn **Time Limit** on, set it to **8** and press **Enter**. Click **Calculate**. Turn **Show Seeds** off and change the **Geomtric COLORING** to **Blue**.



Zones of High Viscosity Surrounding the Small Mixer Blades



14. For **Scalar Function**, click **Select...** On the **Function Selection** panel, select **VISCOSITY** and click **Calculate**.
15. Change the **COLORING** of rake 3 (should be the current rake) to **Scalar**. Change **Time Limit** to **2** and press **Enter**. Click **Calculate**. On the **Colormap** tab, change the **Colormap** to **NASA-1** and adjust the range maximum to **0.45**.
16. Change the current **Rake ID**: to **2**. On the **Rake** tab, change **COLORING** to **Scalar**. Change **Time Limit** to **2** and press **Enter**. Click **Calculate**.
17. Change the current **Rake ID**: to **1**. Turn **Visibility** off.
18. "Quick-pick" (double-click) the outer wall of the mixer (boundary surface 3) and turn **Threshold Clip** off.
19. Using the mouse controls **M1**, **M2**, and **M3**, adjust the view to match that as shown. Observe the highly viscous regions just beyond the tips of both small mixer blades.

Circular Seeding

This **FVX** script is a tool for inserting a circular panel of streamline seeds into a pre-loaded dataset in **FieldView**. Running the script initiates an interactive GUI that provides placement and sizing options to the user. With this GUI, the user can choose the X, Y, and Z coordinates of the seeding circle's center, the direction normal to the panel (X, Y, or Z), the dimensions of the panel, and its bi-directional seed count. The GUI accepts these inputs, creating a rake of streamlines in the dataset.

How to Use Circular Seeding

The simple process of running the circular seeding **FVX** script and the subsequent GUI are outlined in the following steps. Examples of this process are provided in later sections.

1. Load the desired dataset into **FieldView** and visualize any boundary or coordinate surfaces needed to help reference the desired position of the seed circle.
2. Select **Tools** → **FVX Utilities** → **Circular Streamline Seeding**. This will launch the GUI for this streamline seeding tool.
3. The eight input fields on the **Circular Seed Distribution** GUI are initially set to default values. A circular seed panel created using these defaults will be positioned at the origin (center **X**, **Y** and **Z** coordinates are all 0) perpendicular to the X direction having a **Radius** of 1 and a full **Theta** range of 360°. By default, 8 seed rings of 24 seeds each will be created as indicated by the **Radius Count** and **Theta Count** fields, respectively. Ranges of values (minimum and maximum) can be entered into the **Radius** and **Theta** fields to create seeding rings (using **Radius** range), seeding wedges (using **Theta** range) or seeding arcs (using both **Radius** and **Theta** ranges). The seeds placed along the radial and angular directions of the seeding circle (as indicated by the **Radius** and **Theta Counts**) are evenly spaced along their respective directions. Seed circles and seed wedges (with no **Radius** range) also include a seed located at the center point of the circle. If “m” seeds are seeded along the radius and “n” seeds are seeded circumferentially at each radius increment, the total number of seeds produced in the rake is the product of “m” and “n” plus one central seed, provided the circle's center and **Radius** and **Theta** settings generate a seeding panel that lies within the dataset (although this is not required for the script to run).

$$\text{Total Number of Seeds} = m \times n + 1$$

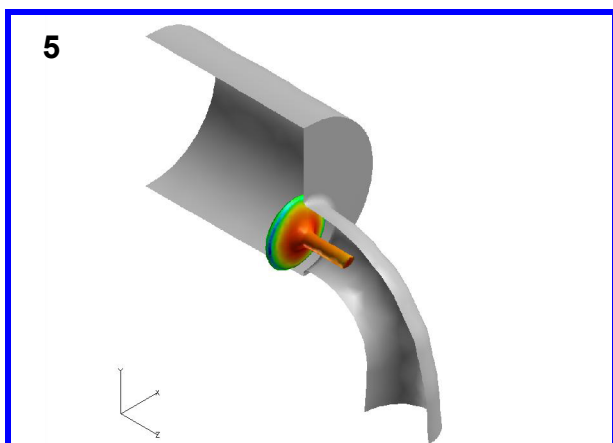
Using default values (m = 8, n = 24) creates a circular panel with 193 seeds.

4. Make any desired changes to the GUI field values. Note that the three possible inputs to the **Set Normal** field are case sensitive (capital letters only). For each change to register correctly with the interface, type the new value into the appropriate field and press **Enter**. As mentioned above, it is acceptable to have the center of the circle or any seeds outside the dataset. When making use of this property, note that seeds lying outside the boundaries of the dataset will not be included in the rake that is produced.
5. Press **Create Streamline Rake** to initiate seed placement and the streamline calculations.
6. The streamline rake along with the panel of seeds should now appear in the **FieldView** graphics window at the appropriate location. Errors, if applicable, are printed in the terminal window from which **FieldView** was launched (Linux systems) or in the MS-DOS window (Windows systems).
7. The **Delete Streamline Rake** button on the GUI can be used at any time to delete the most recent streamline rake. Additional rakes with different characteristics can be created with the GUI by repeating Steps 4 and 5.

Circular Seeding in the Basic Automotive Tutorial Dataset

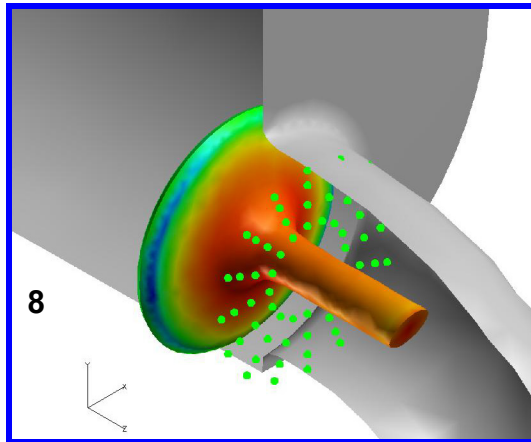
The Basic Automotive Tutorial ([Chapter 5](#)) allows the user to analyze the flow inside a valve intake. Because the cross-sections of the outer wall and the valve are circular, the circular seeding method is ideal for seeding streamline rakes at various points within this geometry. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of **0.5**.

Visualizing the Boundary Surfaces

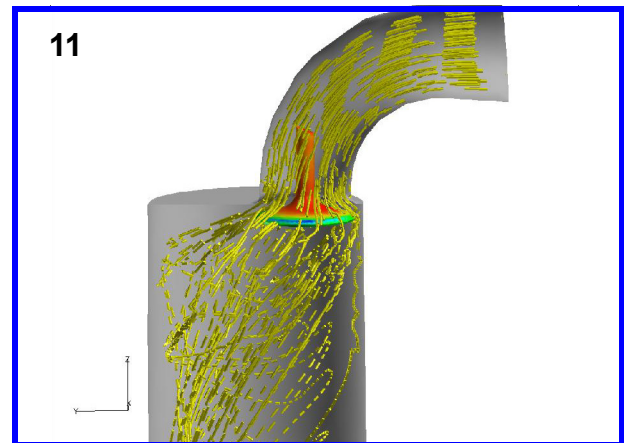
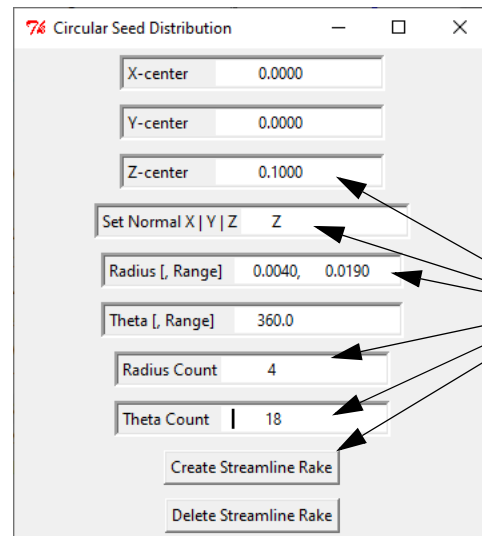


1. Bring up **File** → **Data Input** → **FLUENT [FV-UNS Export]**. Read **Grid or Combined Data intake.uns**. Close the **FLUENT [FV-UNS Export]** panel.
2. To visualize the geometry, click the side toolbar **Bound** icon OR use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. For **BOUNDARY TYPE**, select **wall-8** and click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading.
3. For **Threshold Function**, click **Select...** On the **Function Selection** panel, select **X** and click **Calculate**. Turn **Threshold Clip** on and set **Min:** to **0**.
4. **Create** a second boundary surface with **BOUNDARY TYPE wall-9**. Change the **COLORING** to **Scalar** (the current **Scalar Function** is **density**). Turn **Threshold Clip** off.
5. On the **Transform Controls** toolbar, click **Viewer Options...** Turn **Outline** and **Perspective** off and **Close** the **Viewer Options** panel. Click the **Transform Controls** toolbar icon to **Center** the image.

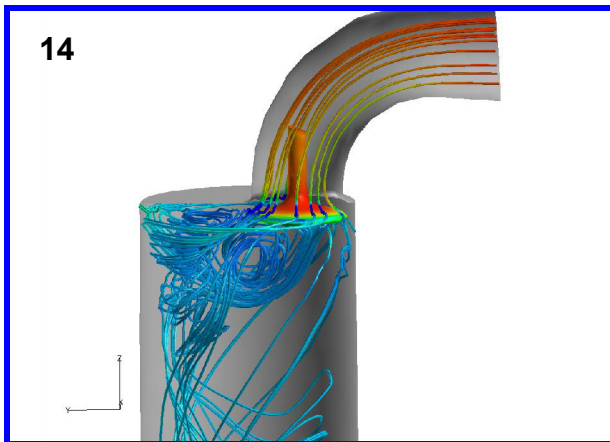
Flow Patterns Around the Valve Inlet



6. Bring up **Tools** → **FVX Utilities** → **Circular Streamline Seeding** to run the script and initiate the GUI.
7. Change **Z-center** to **0.01** and press **Enter**. Change **Set Normal** to **Z** and press **Enter**. This positions the circle perpendicular to the Z-axis just above the main valve disk. To create a seeding ring that extends from the valve shaft to the outer wall, set **Radius [,Range]** to **0.004, 0.019** and press **Enter**. Change **Radius Count** to **4** (and press **Enter**) and **Theta Count** to **16** (and press **Enter**). Click **Create Streamline Rake**.
8. In a few seconds, the seeding circle and streamline rake should appear. On the **Streamlines** panel, turn **Visibility** off. Use right mouse **M3** to zoom in on the seeding panel.
9. Turn **Visibility** back on and turn **Display Seeds** off. In the **Calculation Parameters** section, change the **Direction** to **Both**. Click **Calculate**. Change the **DISPLAY TYPE** from **Complete** to **Filament** and increase the animation divisions **Div:** to **100**. Change the **Geometric COLORING** to **Yellow**.
10. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Click **–X** and **Close**. Change **Action:** to **Locked Rotation** and use **M3** to rotate the model 90° counterclockwise as shown. Reset **Action:** to **Multi-Transform** and use **M2** to rotate the image down slightly. Click the **Transform Controls** toolbar icon to **Center** the image and use **M1** and **M3** to translate and zoom the view as needed.
11. On the **Streamlines** panel, turn **Animate** on to view the flow pattern over the valve.



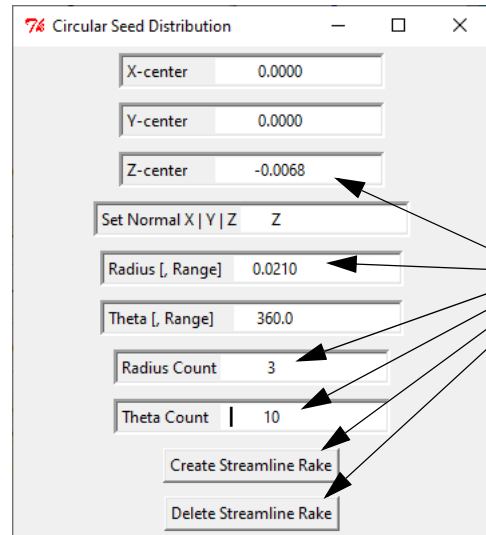
Re-circulating Flow Behind the Valve



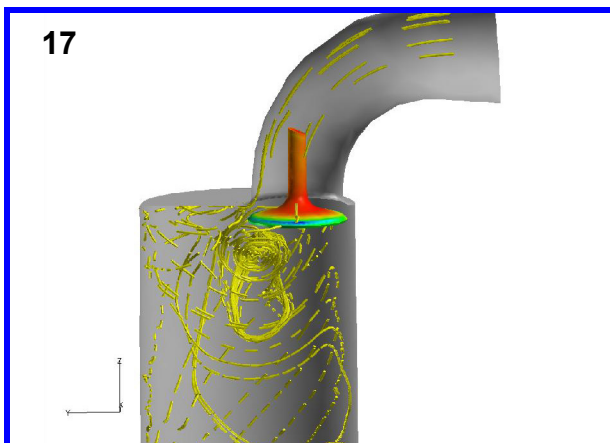
12. On the **Circular Seed Distribution** GUI, click **Delete Streamline Rake**. To seed a circular rake just below the valve, set **Z-center** to **-0.0068**. Change the **Radius** range to a single value of **0.021**. Decrease **Radius Count** to **3** and **Theta Count** to **10**, remembering to press **Enter** after each change. Click **Create Streamline Rake**.
13. For **Scalar Function**, click **Select...** On the **Function Selection** panel, select **pressure** and click **Calculate**. Change **COLORING** to **Scalar** and turn **Show Seeds**

off. In the **Calculation Parameters** section, change the **Direction** to **Both**. Click **Calculate**.

14. On the **Colormap** tab, adjust the minimum spectrum value to **87,000**.

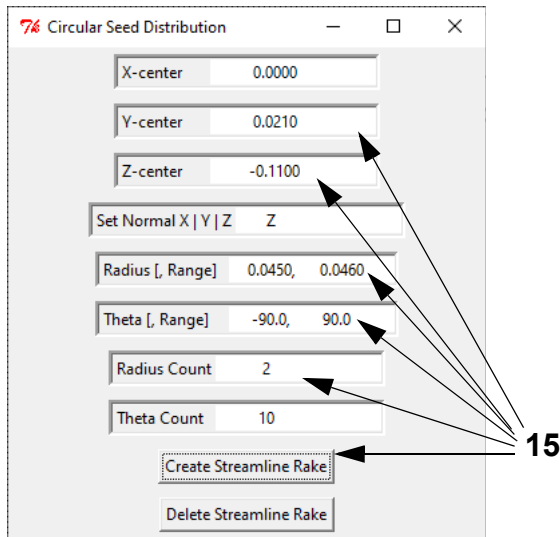


Intake Chamber Perimeter Exit Flow



15. On the **Circular Seed Distribution** GUI, click **Delete Streamline Rake**. To seed a circular rake at the perimeter of the chamber exit, set **Y-center** to **0.021** and change **Z-center** to **-0.11**. Change **Radius [Range]** to **0.045, 0.046** and **Theta [Range]** to **-90, 90**. Reduce **Radius Count** to **2**, remembering to press **Enter** after each change. Click **Create Streamline Rake**.

16. Initially, minuscule streamlines appear because the current seeding arc is near the exit of the chamber and the default streamline **Direction** is **Forward**. On the **Streamlines** panel, change the **Direction** to **Both** and click **Calculate** to trace the streamlines backwards from the seeding locations.

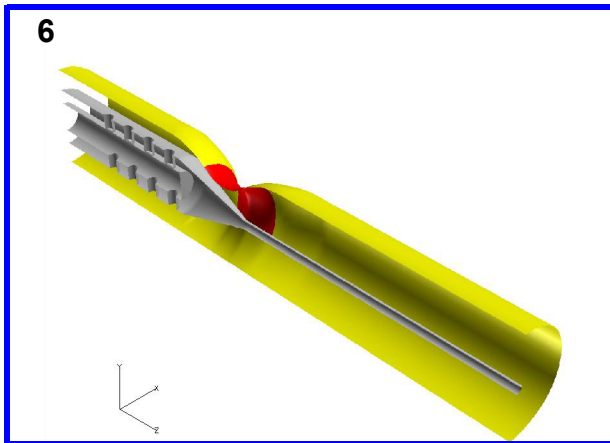


17. Turn **Show Seeds** off and change the **DISPLAY TYPE** to **Filament**. Increase **Div:** to **1000** and turn **Animate** on. Note the zone of heavy re-circulation below the valve and the flow pattern leading to the exit perimeter.

Circular Seeding in the Biomedical Tutorial Dataset

The Biomedical Tutorial ([Chapter 10](#)) investigates the flow field from a catheter injecting contents within an artery. Applying the circular seeding utility allows the user to efficiently seed the circular cross-sections of the catheter and the artery. All images are shown with **Presentation Rendering** on with **SHININESS Intensity** and **Highlight Size** values of **0.5**.

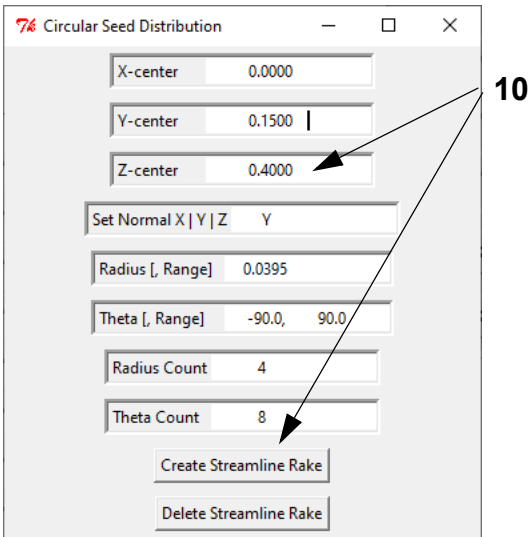
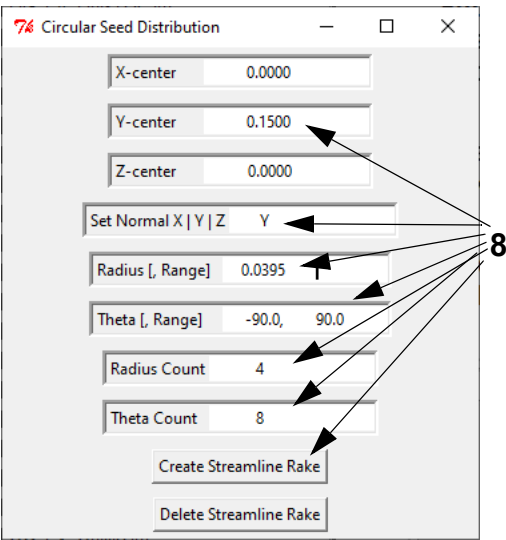
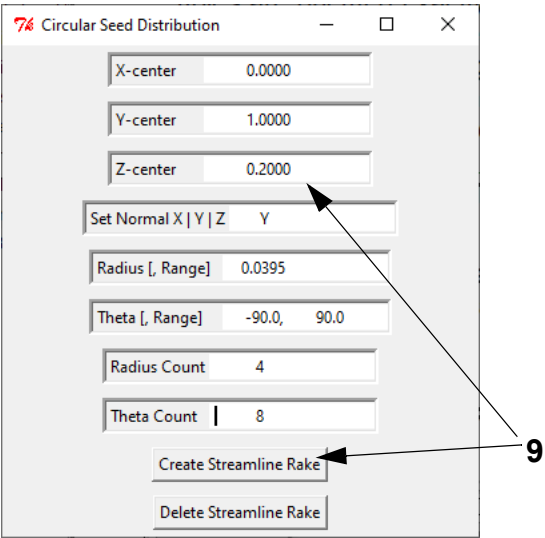
Visualizing the Boundary Surfaces



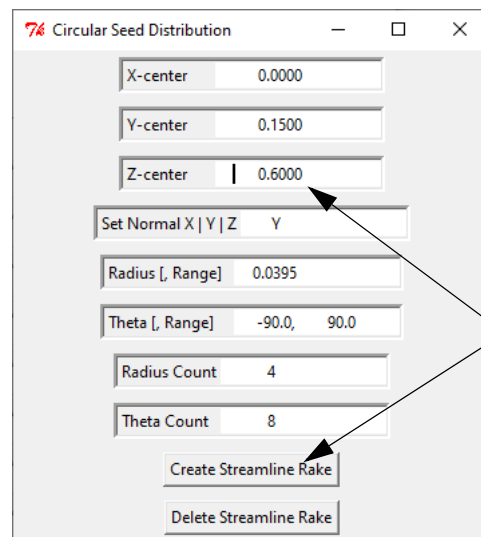
1. Bring up the **File** → **Data Input** → **AcuSolve [FV-UNS Export]** panel. Click **Read Grid or Combined Data...** In the file browser, select **catheter.uns** and click **Open**. **Close** the **AcuSolve [FV-UNS Export]** panel. Note that a second dataset will not be appended here as in the Biomedical Tutorial.
2. Click the side toolbar Bound icon OR use the **Visualization Panels** menu to bring up the **Boundary Surface** panel. Click **Create**. Scroll **BOUNDARY TYPES**, select **tumor** and click **OK**. Change the **DISPLAY TYPE** from **Mesh** to **Smooth** shading. Change the **Geometric COLORING** to **Red**.
3. **Create** a second boundary surface consisting of **artery wall 1** and **artery wall 2**. Change the **Geometric COLORING** to **Yellow**.
4. **Create** a third boundary surface consisting of **wall-1A** through **wall-4A**, **wall-1B** through **wall-4B**, **catheter-I**, **wire** and **catheter-O**. Change the **Geometric COLORING** to **Gray**.
5. On the **Transform Controls** toolbar, click the icon to turn the **Outline** off.
6. Use the mouse controls **M1** and **M3** to produce the view shown.

Visualizing the Catheter Flow

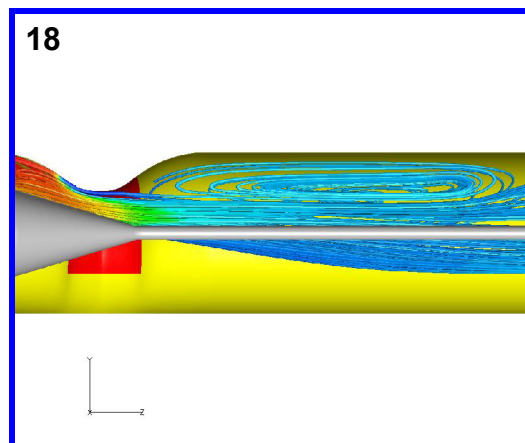
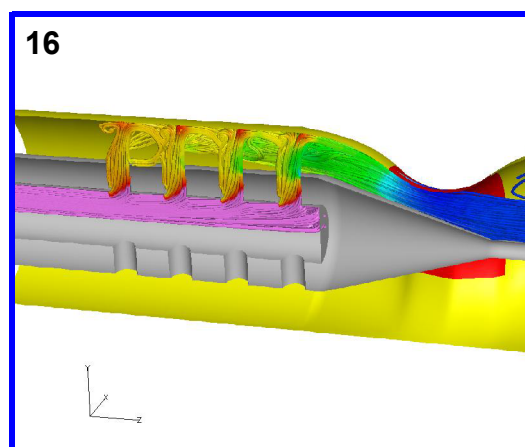
- Bring up **Tools** → **FVX Utilities** → **Circular Streamline Seeding** to run the script and initiate the GUI.
- The catheter administers its drug contents to the artery by means of the four holes located on both the top and bottom of the main catheter body. To demonstrate the effectiveness of the drug distribution from these locations, four streamline rakes will be seeded in the four top holes using the circular seeding method. All of the holes are oriented exactly perpendicular to the Y-axis. Seed a rake of streamlines inside the first hole (top left in the current view) using the following values (press **Enter** after inputting each): Change **Y-center** to **0.15**, **Set Normal** to **Y**, **Radius** to **0.0395** and **Theta** **[,Range]** to **-90, 90**. For even seeding, change **Radius Count** to **4** and **Theta Count** to **8**. Click **Create Streamline Rake**.
- Change **Z-center** to **0.2** and click **Create Streamline Rake** to shift the next streamline seeding semi-circle to the location of the second hole.
- To create the third rake, change **Z-center** to **0.4** and click **Create Streamline Rake**.

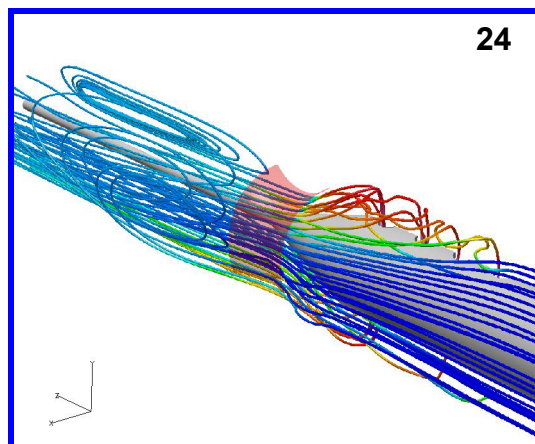


11. To create the final rake, change **Z-center** to **0.6** and click **Create Streamline Rake**.
12. On the **Streamlines** panel, change the **Direction** to **Both** and click **Calculate**. Turn **Show Seeds** off to hide the seed semi-circle.
13. For **Scalar Function**, click **Select...** On the **Function Selection** panel, select **pressure** and click **Calculate**. Change **COLORING** to **Scalar**. On the **Colormap** tab, change the **Colormap** from **Spectrum** to **NASA-1** and the maximum to **270**.
14. The coloring and colormap settings applied in the previous steps alter the appearance of the current rake only (Rake ID 4). Use **Tools** → **Unify** to apply the settings to all four rakes.
15. Though all color settings are uniform, the first three streamline rakes are only plotted in the forward direction and have seed visibility turned on. For each of the three rakes, on the **Rake** tab, change the **Direction** to **Both**, click **Calculate** and turn **Show Seeds** off.
16. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Click **–X** and **Close** the panel. Translate, rotate and zoom in on the holes as shown using the mouse controls **M1**, **M2** and **M3**. Change **Object:** from **World** to **Light** and use **M2** to position the light to better illuminate the featured streamlines. When finished, reset **Object:** to **World**. Note the high-pressure regions on the inner walls of the catheter and the wall of the artery above each hole as well as the distinct circulation zones at these artery wall locations.
17. Change the **Scalar Function** from **pressure** to **species conc 1** to observe the drug concentration over the streamline paths. Use **Tools** → **Unify** to apply this setting to all four rakes.
18. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Click **–X** and **Close** the panel. Click the **Transform Controls** toolbar icon to **Center** the image. Use **M1** and **M3** to zoom in on the portion of the artery behind the tumor. Observe the region of high circulation in this section.

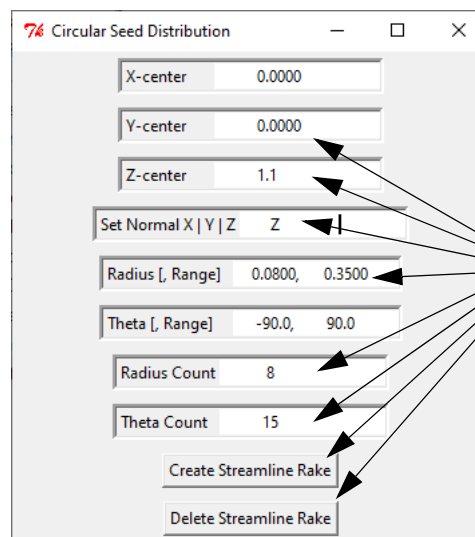


11



Flow Through the Tumor Region

19. On the **Circular Seed Distribution** GUI, click **Delete Streamline Rake** four times to delete the four streamline rakes. Change **Y-center** to **0**, **Z-center** to **1.1**, **Set Normal** to **Z**, **Radius [, Range]** to **0.08, 0.35**, **Radius Count** to **8** and the **Theta Count** to **15**. Click **Create Streamline Rake** to seed the cross section of the artery at the location of the tumor.
20. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Choose **ISOMETRIC** direction **-X +Y +Z** and **Close** the panel. Observe the location of the seeding plane. The range of radius values is utilized here to seed only between the catheter tube and artery wall surfaces. Note that the outer radius value of **0.35** is the radius of the non-affected artery. Though the portion affected by the tumor (top portion) has a smaller radius the seeds of the semi-circle lying outside this boundary are not considered.
21. On the **Streamlines** panel, change the **Direction** to **Both** and click **Calculate**. Change **COLORING** to **Scalar (Scalar Function** should still be **species conc 1** from step 17.). Turn **Show Seeds** off.
22. “Quick-pick” (double-click) the yellow artery wall boundary surface and turn its **Visibility** off. “Quick-pick” the red tumor boundary surface and adjust its **Transparency:** to **62.5%**.
23. Click the **Transform Controls** toolbar icon to bring up the **Defined Views** panel. Change the **ISOMETRIC** view to **+X +Y -Z** and **Close** the panel. Click the **Transform Controls** toolbar icon to **Center** the image.
24. Using the mouse controls **M1** and **M3**, adjust the view to match that as shown.



Chapter 16

16

Working with XDBs Tutorial

What You Will Learn

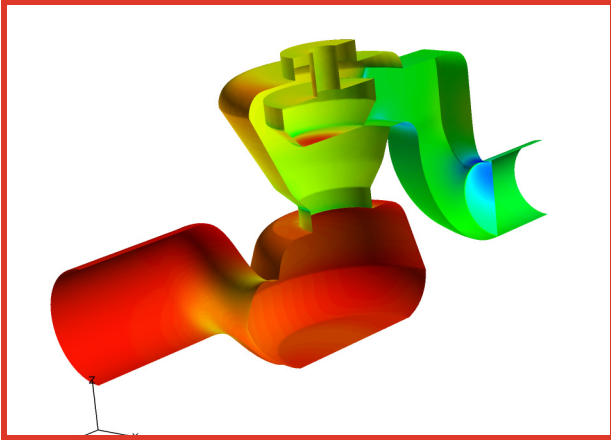
- [How to create an XDB dataset](#)
- [How to import an XDB dataset](#)
- [How to use XDB datasets for Dataset Comparison](#)
- [How to perform numerical dataset comparison](#)

Step 1: Create an XDB dataset

In this example, we will create an XDB dataset containing boundary surfaces and multiple coordinate planes. All of the data stored in XDB datasets is 'surface' only; no volume data is saved.



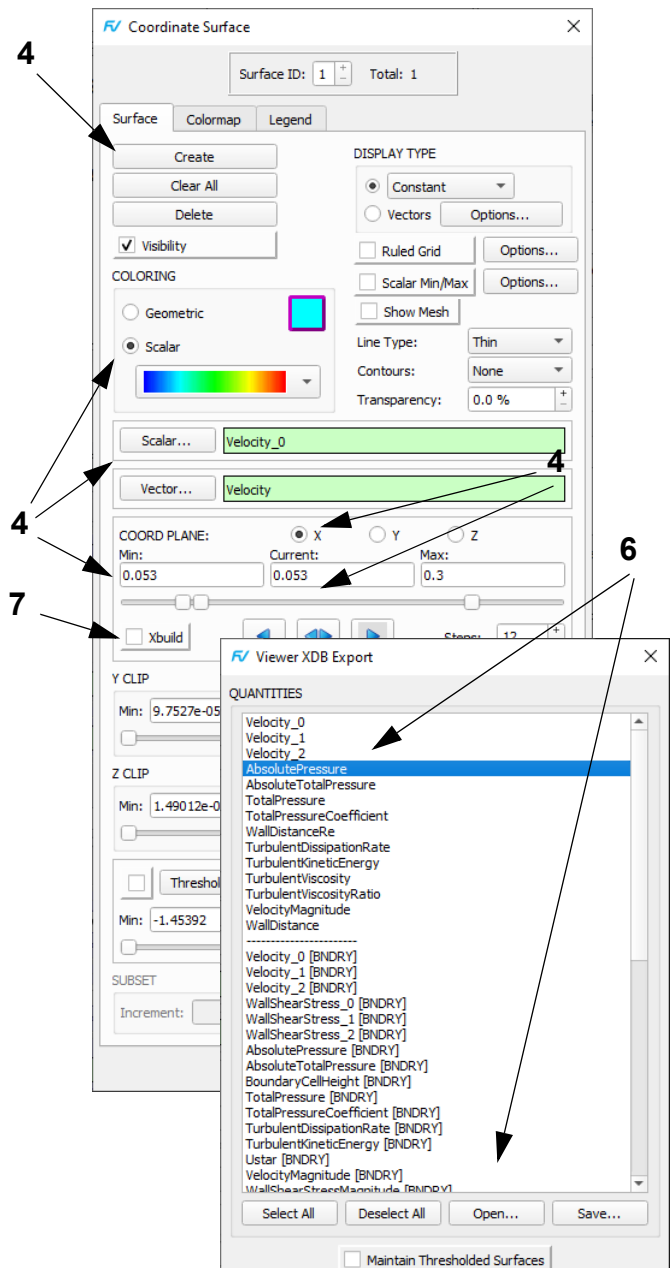
The pictures on each page correspond to the final image you should see when the steps are completed correctly.



1. Bring up the **File→Data Input→FLUENT [FV-UNS Export]** data reader, click on the **Read Grid or Combined Data...** button, choose **control_valve_arb_poly.fvuns** and click **Open**. Close the **FLUENT [FV-UNS Export]** panel.
2. Click the **Bound** icon on the side toolbar to bring up the **Boundary Surface** panel. Click **Create**, select **valve** from the **BOUNDARY TYPES** list, then click **OK**. Change the **DISPLAY TYPE** to **Smooth** shading, change **COLORING** to **Scalar** and set the **Scalar Function** to **AbsolutePressure**.
3. **Create** a second boundary surface, this time selecting **wall** from the **BOUNDARY TYPES** list.
4. Click the **Coord** icon on the side toolbar to bring up the **Coordinate Surface** panel. **Create** an **X** coordinate surface. Set the **Min:** and **Max:** sweep extents in the **COORD PLANE:** to be 0.053 and 0.3, respectively. Set the **Current** position to 0.053. Change the number of **Steps** from the default of 25 to 12. Change the **DISPLAY TYPE** to **Constant** shading, set **COLORING** to **Scalar** and select **Velocity_0** as the **Scalar Function**.
5. Use the Transform Controls toolbar icon or the View pulldown menu to toggle the **Outline** display **Off**. Transform the model to correspond to the image shown.
6. Go to **Tools→Viewer XDB Build Mode**. In the **Viewer XDB Save** file browser, specify the name **multiple_X_coords** and click **Save**. On the **Viewer XDB**

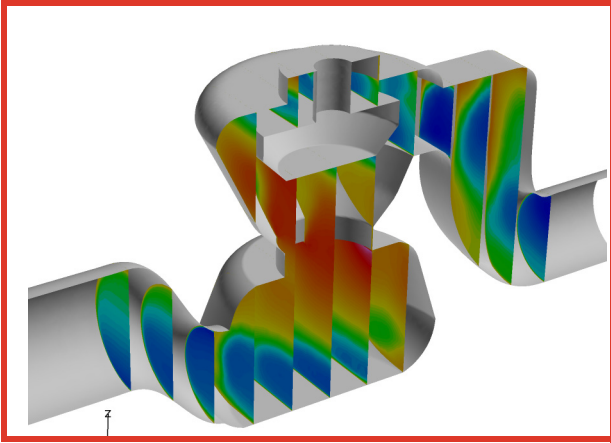
Export panel, select the **QUANTITIES AbsolutePressure** and **Velocity**; click **OK** at the bottom of the panel (OK button, and velocity function are off screen here.)

7. On the **Coordinate Surface** panel, turn on **Xbuild** to create your XDB dataset.

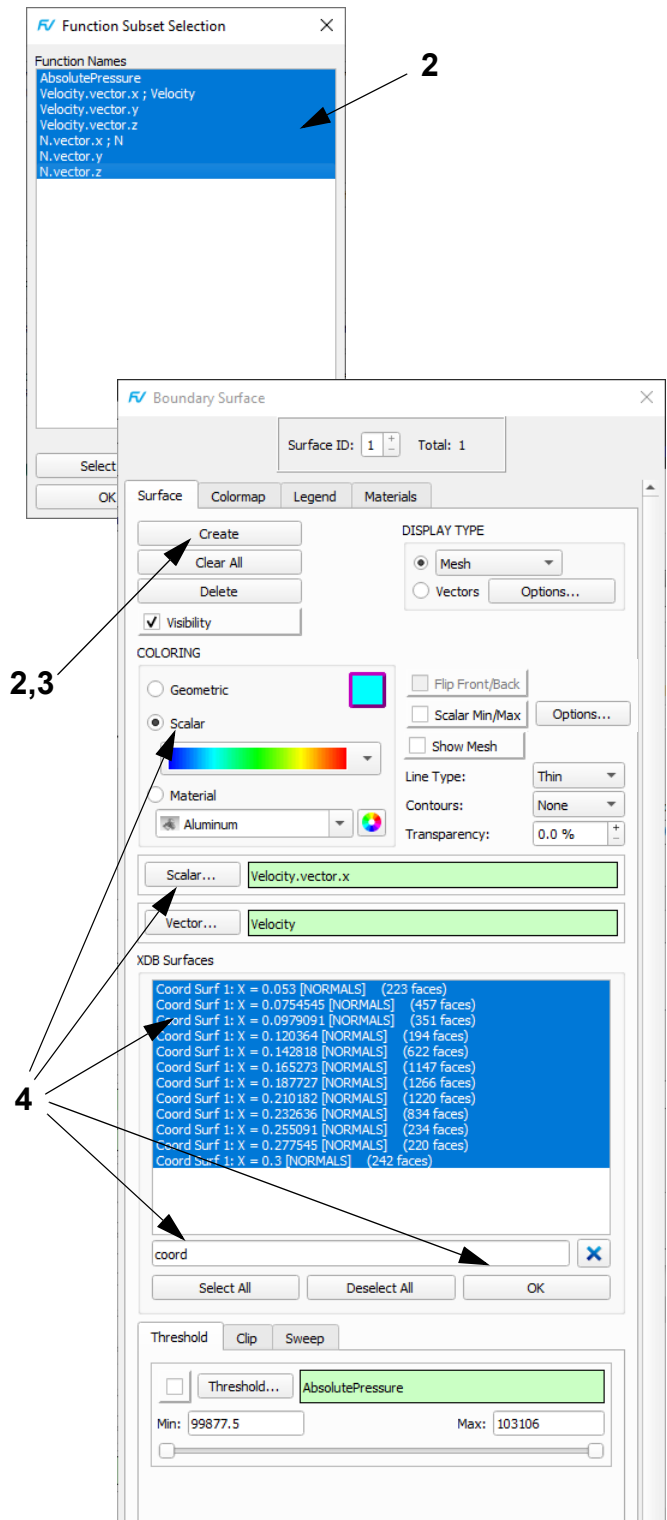


Step 2: Import the XDB dataset and display the results

In the previous step, an XDB dataset was created. At the same time, a Complete Restart was saved which can be used to re-create the surfaces and rakes using the original volume data.

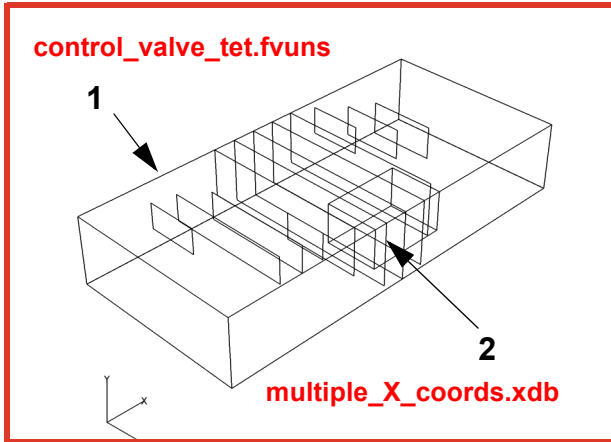


1. Bring up the **File→Data Input→XDB Import** data reader, click on the **Read Grid & Results Data...** button, choose **multiple_X_coords.xdb** and click **Open**.
2. When the **Function Subset Selection** panel comes up, note that the scalar function **AbsolutePressure** and the vector components of **Velocity** appear in the Function Names list. These functions were saved during XDB export in **multiple_X_coords.xfn**. Surface normals, vector quantity **N**, have also been saved. Click **OK** on the **Function Subset Selection** panel. **Close** the **XDB Import** panel.
3. Bring up the **Boundary Surface** panel and click **Create**. All surfaces stored in the XDB dataset are Boundary Surfaces. The naming convention gives an indication of how the surfaces were derived. The original boundary surfaces for the valve and wall are at the top of the **BOUNDARY TYPES** list. Select both and click **OK**. Change the **DISPLAY TYPE** to **Smooth**.
4. Each **Coord Surface** generated during the **Xbuild** sweep has its X-location included in its name. **Create** a new **Boundary Surface**, select all **BOUNDARY TYPES** with the **Coord Surf** entries (using the filter field) entries and click **OK**. Change **COLORING** to **Scalar** and change the **Scalar Function** to **Velocity.vector.x**. To recreate the view shown, bring up **File→Open Restart→View (World)**, select **multiple_X_coords_xdb.vct** and click **Open**.



Step 3: Using XDB datasets for Dataset Sampling

The use of XDB datasets for Dataset Sampling is an order dependent operation. We will use a full volume dataset as the 'Results Target' and the XDB dataset from the previous step as the 'Grid Target'.



1. Toggle the **Outline** display **On**. Bring up the **File→Data Input→FV-UNS** data reader, make sure that the **INPUT MODE** is set to **Replace**, click on the **Read Grid or Combined Data...** button, choose **control_valve_tet.fvuns**, and click **Open**. Close the **FV-UNS** panel. This is your 'Results Target'.
2. Bring up the **File→Data Input→XDB Import** data reader, change the **INPUT MODE** from **Replace** to **Append**, click on the **Read Grid & Results Data...** button, choose **multiple_X_coords.xdb** and click **Open**, **OK** and **Close**. This is your 'Grid Target'.
3. Go to **Tools→Dataset Sampling** and select Dataset #1 in the flyout menu to identify this as the 'Results Target'. In the **Create Sampled Data** file browser, specify the name **tet_XDB_sampled.uns** and **Save**.

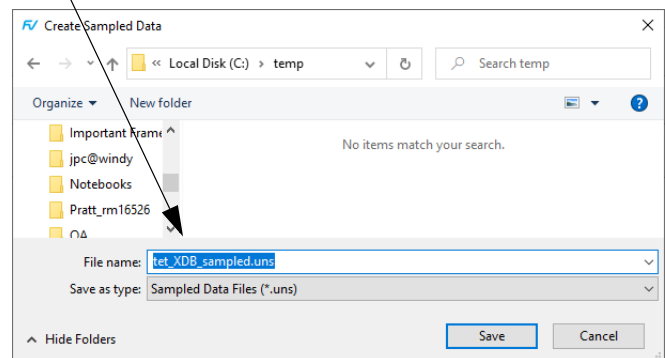
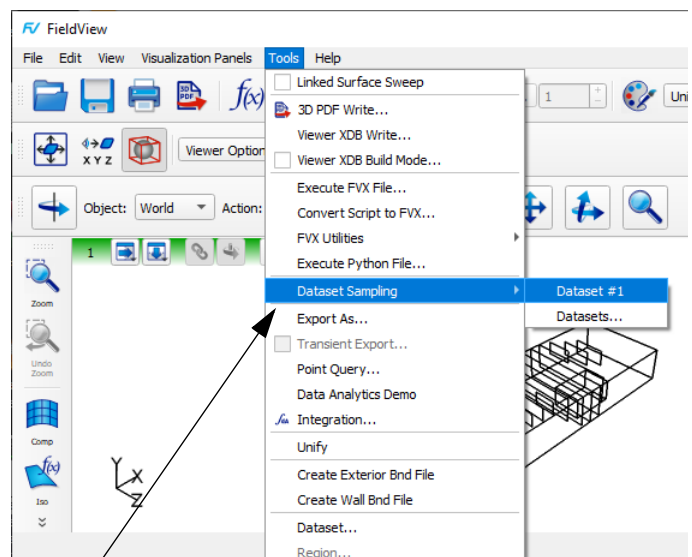
It is possible to automate these steps using **FVX**:

```
read_dataset( {
  data_format = "unstructured",
  input_parameters = {
    name = "control_valve_tet.fvuns",
    options = {
      input_mode = "replace",
      boundary_only = "off"
    } -- options
  } -- input_parameters
} ) -- read_dataset
```

```
read_dataset( {
  data_format = "xdb_import",
  input_parameters = {
```

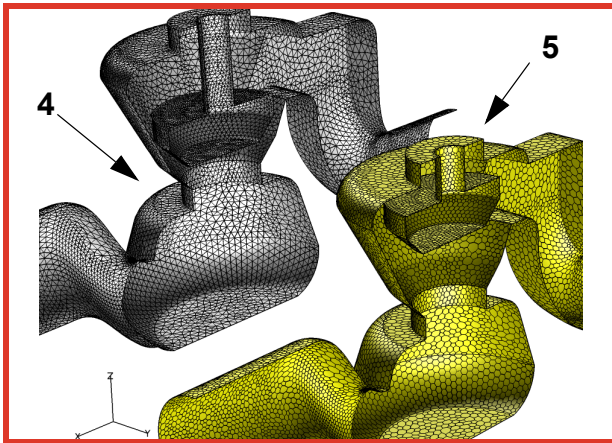
```
name = "multiple_X_coords.xdb",
options = {
  input_mode = "append",
  boundary_only = "off"
} -- options
} -- input_parameters
} ) -- read_dataset
```

```
fv_script('DATASET_SAMPLING 1
tet_XDB_sampled.uns')
```

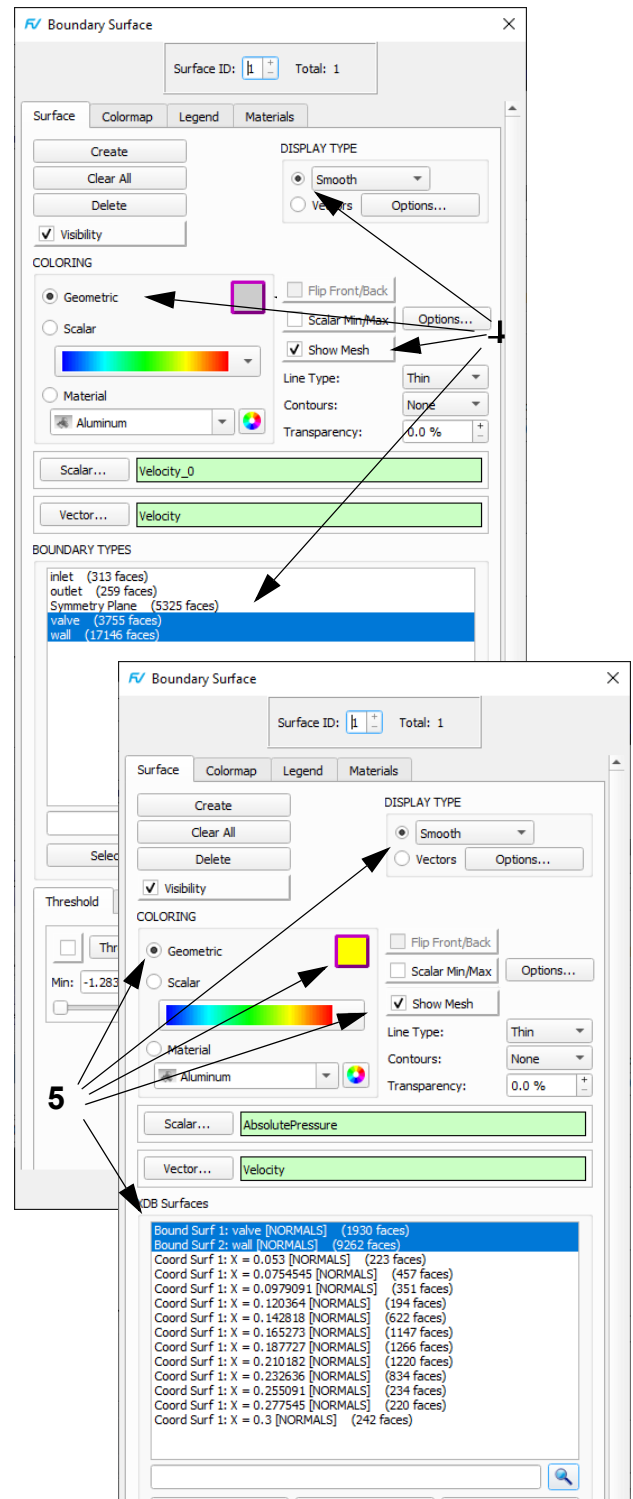


Step 3 (cont'd): Using XDB datasets for Dataset Sampling

We will illustrate surfaces contained within the 'Results Target' and the 'Grid Target' in this step.

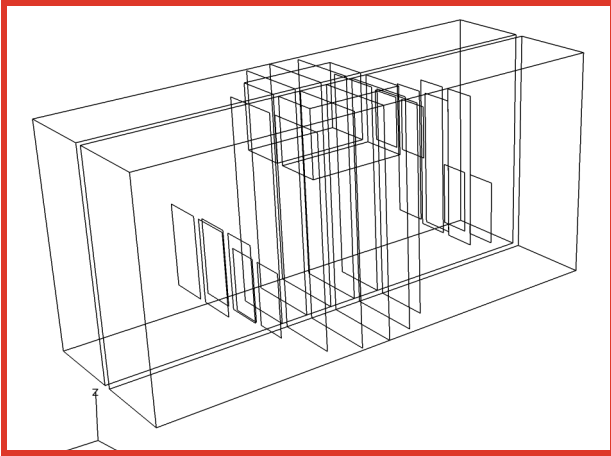


4. Use the main toolbar selector to the right of the **Dataset...** button (or click **Dataset...** to bring up the **Dataset Controls** panel) to change the current dataset to #1. On the **Boundary Surface** panel, click **Create**. From the **BOUNDARY TYPES** list, select **valve** and **wall**, then click **OK**. Change the **DISPLAY TYPE** to **Smooth** shading, leave **COLORING** set to **Geometric**. Turn on **Show Mesh**.
5. Change the current dataset to #2. On the **Boundary Surface** panel, click **Create**. From the **BOUNDARY TYPES** list, select **Bound Surf 1: valve [NORMALS]** and **Bound Surf 2: wall [NORMALS]**, then click **OK**. Change the **DISPLAY TYPE** to **Smooth** shading, leave **COLORING** set to **Geometric**, and change the color chip from gray (default) to yellow. Turn on **Show Mesh**.
6. At this point, the two datasets will be positioned in the same location. On the Viewer toolbar, the **Object** is currently set to **World**. Using the mouse, rotate the **World** view so that the valve is visible in the upper part of the datasets. Translate both datasets to the upper left quadrant of the graphics window. On the Viewer toolbar, change the **Object** from **World** to **Dataset**. Use the mouse to position **Dataset #2** in the lower right quadrant of the graphics window. Toggle the **Outline** display **Off**. On the Viewer toolbar, change the **Object** from **Dataset** back to **World**.



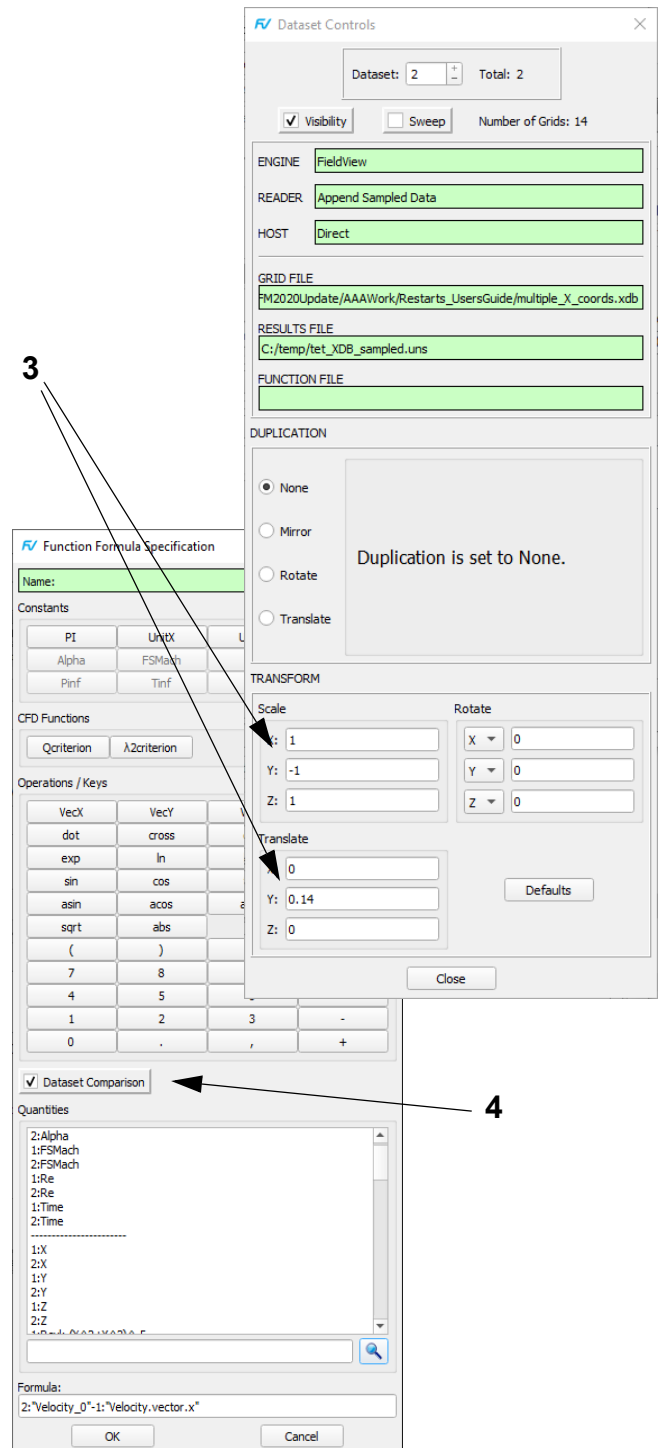
Step 4: Performing Dataset Comparison with the Sampled Data

In this step we will read the XDB dataset and append the sampled data to it so that numerical dataset comparisons can be done.



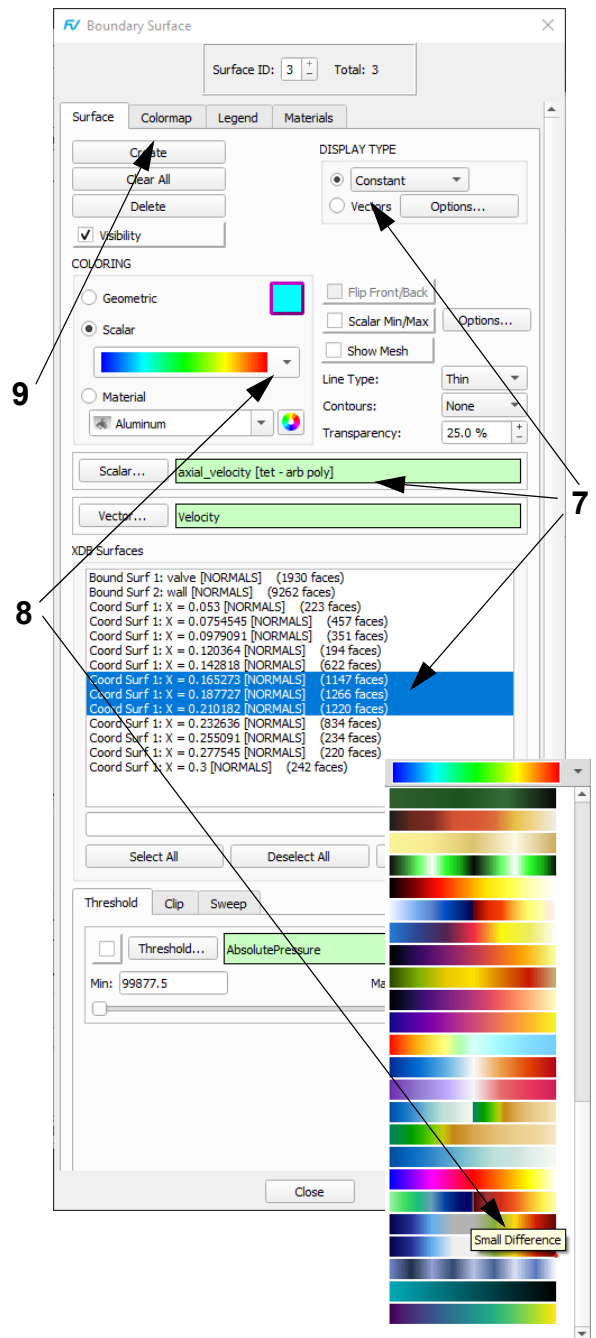
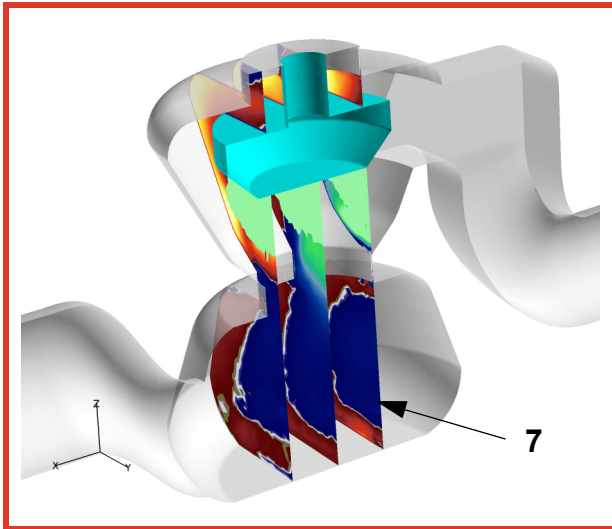
Comparison on to prepend the formulas for both datasets with their respective dataset numbers.

- Bring up the **File→Data Input→XDB Import** data reader, change the **INPUT MODE** from **Append** to **Replace**, click on the **Read Grid & Results Data...** button, choose **multiple_X_coords.xdb** and click **Open**, **OK** and **Close**. This file contains results from the **control_valve_arb_poly.fvuns** dataset. To recreate the view shown, bring up **File → Open Restart → View (World)**, select **multiple_X_coords_xdb.vct** and click **Open**. Toggle the **Outline** display **On**.
- To append the Sampled Data from the previous step, navigate to **File→Data Input→Append Sampled Data**. In the **Append Sampled Data** file browser, select the file **tet_XDB_sampled.uns** and click **Open**. This file contains results from the **control_valve_tet.fvuns** dataset. In comparing the two datasets, note that one has been modeled using arb polys and the other using tets. Also, the valve is located in a different position for each case.
- To show the two datasets side-by-side, we will apply a scale and translation to the Sampled Dataset. Navigate to **Tools→Dataset...** or click the main toolbar **Dataset...** button to bring up the **Dataset Controls** panel, and confirm you are showing **Dataset: 2**. Change the **TRANSFORM SCALE** for **Y** to **-1**, and change the **TRANSLATE** for **Y** to **0.14**. and **Close** the **Dataset Controls** panel.
- In preparation to show numerical differences between the two datasets, we will create difference formulas. Click the main toolbar button or navigate to **Edit→Function Specification...** to bring up the **Function Specification** panel. Click **Create...** On the **Function Formula Specification** panel, turn **Dataset**



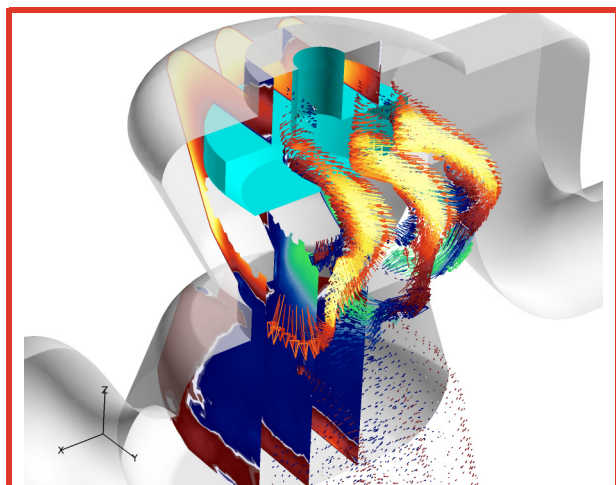
Step 4 (cont'd): Performing Dataset Comparison with the Sampled Data

We will finalize the Dataset Comparison illustration with these steps.

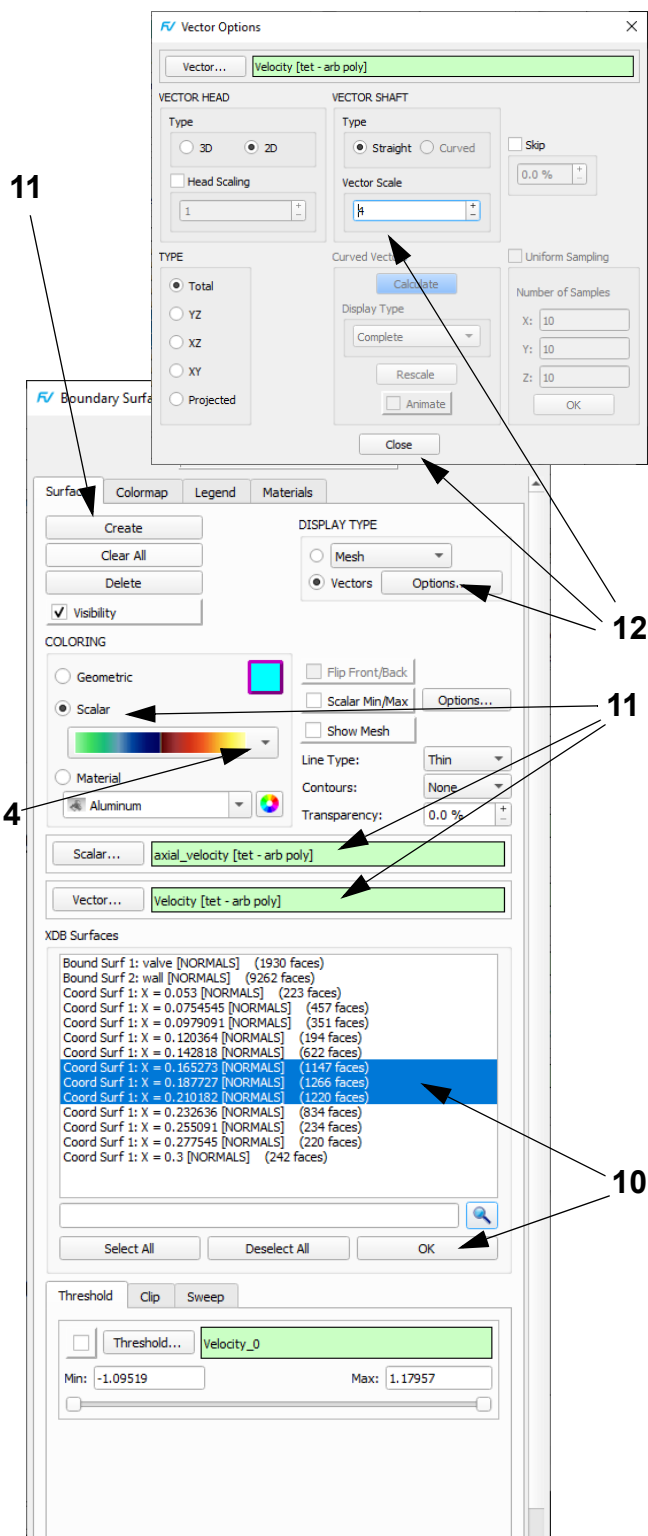


5. In the **Formula:** box, create the formula 2:"Velocity_0"-1:"Velocity.vector.x" and click **OK**. On the **Function Name Input** panel, type in **axial_velocity [tet - arb poly]** and click **OK**. Create a second formula, 2:"Velocity"-1:"Velocity", and name it **Velocity [tet - arb poly]**. Close the **Function Specification** panel.
6. Change the current dataset to #1. On the **Boundary Surface** panel, click **Create**. Select the **BOUNDARY TYPE** named **Bound Surf 2: wall [NORMALS]** and click **OK**. Change the **DISPLAY TYPE** to **Smooth**, leave the **COLORING** at **Geometric**, and change the **Transparency** to 25%. Click on the **Geometric** color chip and change to gray. Create a second Boundary surface using the **BOUNDARY TYPE** named **Bound Surf 1: valve [NORMALS]**. Click on the **Geometric** color chip and change from gray to cyan.
7. Create a third Boundary Surface selecting three **Coord Surf 1: BOUNDARY TYPES** for **X = 0.165273**, **X = 0.187727**, and **X=0.210182**. Change the **Scalar Function** to **axial_velocity [tet - arb poly]**, change **COLORING** from **Geometric** to **Scalar** and set the **DISPLAY TYPE** to **Constant**.
8. To more effectively show differences on the Coord planes, we will use a custom colormap. Click on the Colormap pulldown, and select the **Small Difference** colormap.
9. Open the Colormap tab and change the upper and lower **SCALAR COLORING** limits to 0.5 and -0.5, respectively.

Step 4 (cont'd): Performing Dataset Comparison with the Sampled Data



10. Change the current dataset to #2.
11. On the **Boundary Surface** panel Surface tab, click **Create**. Select the same three **Coord Surf 1: BOUNDARY TYPES** for **X = 0.165273**, **X = 0.187727**, and **X=0.210182** that were used for the third Boundary Surface in Dataset #1.
12. Change the **Scalar Function** to **axial_velocity [tet - arb poly]** and the set the **DISPLAY TYPE** to **Vectors**. Change the **Vector Function** to **Velocity [tet - arb poly]**. Change the **COLORING** to **Scalar**.
13. Click the **Options...** button next to **Vectors** to bring up the **Vector Options** panel. On the **Vector Options** panel, change the **Vector Scale** from 1 (default) to 4. **Close** the **Vector Options** panel.
14. Again, as with dataset #1, click on the Colormap pull-down, and select the **Small Difference colormap**.
15. Rotate the model to examine the differences between the two datasets.



Symbols

%%N1	131
%%T1	131

Numerics

2D Plotting	108
-------------------	-----

A

Add Between	160
Align 95, 97, 105, 112, 114, 125, 180, 226,	228, 229
Animation	
Streamlines & Streaklines .106, 117, 123,	131, 132, 139, 232
Titles	131
Annotation	124, 140
Appending Data	180

B

-batch	9, 10, 13
batch mode	9, 10, 13
Boundary Surface Panel .103, 112, 120, 138,	150, 158, 177, 185, 226
Build Mode	131, 132, 144, 184

C

Calculation Parameters	
Time Limit	123, 160
Calculator .108, 118, 121, 124, 129, 134	
CFD Calculator 108, 118, 121, 124, 129, 134	
CFX-4	15, 53
Data Input	158, 185
dmp files	53
fvbnd files	20
Client-Server Mode	28
Color Mixer	120, 138, 155
Colormap Specification Panel	52
command-line switches	9
-batch	9, 10, 13
-create_wall_fvbnd	9, 14
-create_exterior_fvbnd	9, 14
-f	9, 10, 12
-fc (Fast)	9, 16
-fvb	11
-fvx	12

-gamma	10, 16
-gasconstant	10, 16
-pN	9, 10, 14, 133
-s	9, 10, 12, 13, 15
-sb	9
-silent	10
-size	10, 17

Complete Restart	54
Computational Surface Panel97, 123	
Contour Lines	125
Contours	
Filled	114, 140, 153, 228, 229
Contours as attributes	114, 228, 229
Control Panels	
Colormap Specification Panel	52
Dataset Controls	
Scaling 112, 134, 150, 226	
Transient Data Controls Panel	15
Convective Flux	154
Coordinate Surface Panel	105, 159
Coordinate Surface Restart	140
Create Wall Bnd File	112, 226
-create_wall_fvbnd	9, 14
-create_exterior_fvbnd	9, 14

D

Data	
Appending	180
CFX-4	158, 185
Duplication	120
FLOW-3D®	95
FV-UNS	125, 138, 150, 177
Mirroring	95, 154
PLOT3D	14, 15, 16, 18, 20, 120
Q variables 15, 22	
RAMPANT/FLUENT-UNS	103
Rotational copies	120
Transient	13, 15, 99, 100, 125
DataGuide	13, 14, 15, 16
Grid Subsetting	14
Point Subsetting	14
Dataset Controls	
Negative Scaling .112, 134, 150, 226	
Reflection	112, 134, 150, 226
Scaling	112, 134, 150, 226
Dataset Controls Panel	181

Dataset Reflection	112, 134, 150, 226
Diffusive Flux	154
Display Options	
Contours	114, 228, 229
Transparency	112, 120, 150, 154, 183, 226
dmp files	53
droplet	177

E

Eigenmode Analysis	180
Exploded view	138
Exporting Streamlines	131

F

Fast mouse controls	9, 16
Fast Viewing Controls	9, 16
-fc	9, 16
Feature Extraction	
Separation / Reattachment	177
Surface Flows	177
Vortex Cores	177
Eigenmode Analysis	180
Vorticity Alignment	177
FIELDVIEW-UNS	14, 125, 138, 150, 177
Filled Contours	114, 140, 153, 228, 229
Flipbooks	99, 107, 131, 144, 184
Build Mode	131, 132, 144, 184
Flooded Contours	140, 153
Flow Number (FI)	162
FLOW-3D®	
Data Input	95
Flux	
Convective	154
Diffusive	154
Formula Restart	141, 151, 159
FORTTRAN language	
Files	
Formatted	13
Function Formula Specification	108, 118, 121, 124, 129, 134
Function Selection	77
Function Specification Panel	
Registers	52
Scalar	52
Threshold	52

Vector 14, 52

Functions

PLOT3D	14, 15, 16, 18, 20
fv	
-batch	9, 10, 13
Command Line Switches	9
-create_exterior_fvbnd	9, 14
-create_wall_fvbnd	9, 14
-f	9, 10, 12
-fc (Fast)	9, 16
-fvb	9, 11
-fvx	9, 12
-gamma	10, 16
-gasconstant	10, 16
-pfvN	9
-pN	9, 10, 14, 133
-python	9
-s	9, 10, 12, 13, 15
-silent	10
-size	10, 17
fv -port none	33
FV_2D_TO_3D	18
FV_ALT_LOOKUP	18
FV_ARB_POLY	18
FV_DATA_CHECK	18
FV_DEBUG_AUTOSTART	19
FV_DEFAULT_PORT	33
FV_DP_LAYERS	23
FV_HOME	1, 24
FV_IGNORE_WALLS	19
FV_MULTISAMPLE_ENABLE	20
FV_NO_BNDRY_FILE	20
FV_NO_DATA_CHECK	20
FV_NO_GRID_JUMP	20
FV_NO_SCONFIG_ECHO	20
FV_NO_STREAK	20
FV_OLD_BROWSER_BEHAVIOR	20
FV_PLUGINS	21
FV_PRINT_DIR	21
FV_PRINT_NO_AA	21
FV_RF_GRAPHICS	21
FV_SCREEN_GRAB	21
FV_SHOCK_MIN_NODES	22
FV_SHOCK_PRESSURE_CHANGE	22
FV_SHOW_WALLS	22
FV_SINGLE_FILE_STREAKLINES	22

FV_SSLINES_EXPORT	23
FV_USE_LONGEST_PATH	24
fvbnd	112, 226
FV-UNS	14, 125, 138, 150, 177
Data Input	125, 138, 150, 177

G

-gamma	10, 16
gamma	10, 16
-gasconstant	10, 16
gasconstant	10, 16
Geometric Streamline Seeding	199
Graphics Layout Size (NTSC, PAL, D1)	183
Grid Subsetting	
DataGuide	14
PLOT3D	14

H

Heat Flux	118
-----------------	-----

I

Impeller	162
Importing Particle Paths	132
Integration Controls Panel	110, 141, 142, 154, 162
Integration Direction	181
Integration Step	181
Isometric View ..	112, 114, 226, 228, 229
Iso-Surface Panel	95, 121, 139, 162

K

Keyframe Animation	143, 183
Frame Display	143
Graphics Layout Size (NTSC, PAL, D1)	143, 183
Time Line	144, 145, 184
Track Selection	143, 183
Keyframe Restart	144, 145, 183

L

Legend transform	115, 117, 230, 232
Legend/Label Controls panel	140
Light transform	114, 228, 229

M

Mass Balance	154
MIFF	134, 144, 184
Animation	131, 134, 144
Mirroring data	95, 154

N

Negative Scaling	112, 134, 150, 226
------------------------	--------------------

O

Outline	95, 103, 112, 120, 150, 158, 177, 185, 226
Overlapping Surfaces	155

P

Parallel Timing Instrumentation	45
Particle Paths	
Importing	123, 132
Residence Times	123
Perspective	113, 227
PLOT3D	14, 15, 16, 18, 20
Data	
XYZ files	14
Data Input	120
PLOT3D/Fast mouse controls	9, 16
-pN	14
pN	14
Point Subsetting	
DataGuide	14
PLOT3D	14
Postscript	19
Preference Restart	12
Presentation Quality	161
Presentation Rendering ..	107, 117, 122, 138, 161, 180, 232
Shine ..	107, 117, 122, 138, 161, 232
Specular Shading ..	107, 117, 122, 138, 161, 232
Printing	
DPI	19
PS	19
Pumping Capacity (Q)	163

Q

Quick-Picking ..	100, 110, 114, 121, 141, 151, 152, 153, 155, 180, 228, 229
------------------	--

R**RAMPANT/FLUENT-UNS**

Data Input 103

Recirculation 151

RenderingPresentation Rendering 107, 117, 122,
..... 138, 161, 180, 232

Shine .. 107, 117, 122, 138, 161, 232

Specular Shading 107, 117, 122, 138,
161, 232

Residence Times 123

Restart

Coordinate Surface 140

Formula 141, 151, 159

Keyframe 144, 145, 183

Restart Files

Complete 54

Preference 12

Rotational copies of data 120

Running multiple FieldViews 2

Rushton Impeller 162

S

Scaling 112, 134, 150, 226

Scaling (vectors) 98, 100, 105, 151, 159

Script Language 10, 12, 13

Seed A Surface 130

Separation / Reattachment 177

Shear Rate 141, 152

Shine 107, 117, 122, 138, 161, 232

-size 10, 17

Space Bar Viewer toggle 181

Specular Shading 107, 117, 122, 138, 161,
..... 232

Streamlines & Streaklines 20

Animation ... 106, 117, 123, 131, 132,
139, 232

Display Types 139

Exporting 123, 131

Integration Direction 181

Integration Step 181

Presentation Quality 161

Presentation Rendering 107, 117, 122,
..... 134, 161, 232

Residence Times 123, 160

Seeding 130, 160, 181

Circular 215

Rectangular 208

Spherical 200

Time Limit 123, 130, 160

Structured boundary file (fvbnd) 20

Subsetting 113, 121, 123, 227

Surface Flows 177

Surface transform 138

TThreshold Clip . 129, 139, 158, 162, 180,
185Thresholding 97, 101, 103, 108, 129, 139,
158, 162, 180, 185

Time Limited Streamlines 123, 130, 160

Time Line 144, 145, 184

Titles 124, 140

Transient 131

Tools

Color Mixer 120, 138, 155

Create Wall Bnd File 112, 226

Graphics Layout Size 183

Transient Data Controls Panel 13

Unify 141, 152, 153

Transform Controls

Legend 115, 117, 230, 232

Options

Perspective 113, 227

Surface 138

World 115, 117, 230, 232

Transient Data 125, 128

Sweeping 128

Transient Data Controls Panel 13, 15, 99,
100Transparency .. 112, 120, 150, 154, 183,
226**U**

Unify 141, 152, 153

V**Vectors**

Scaling 98, 100, 105, 151, 159

ViewAlign 95, 97, 105, 112, 114, 226, 228,
229

Visualization Panels

Boundary Surface Panel	103, 112, 120, .138, 150, 158, 177, 185, 226
Computational Surface Panel	97, 123
Coordinate Surface Panel ..	105, 159
Iso-Surface Panel .	95, 121, 139, 162
Streamlines	20
Vortex Cores	177
Eigenmode Analysis	180
Vorticity Alignment	177
Vorticity Alignment	177

W

WIND	20
fvbnd files	20
Variables	20
window size	10, 17
World transform	114, 115, 117, 228, 229, 230, 232

Z

Zoom Box	122, 139, 151, 159
----------------	--------------------