

RhinoCFD Powered by PHOENICS

RhinoCFD Tutorial - Flow Past a Sphere

Document release date: April 2023 Software version: 2.1.5 January 2022 Solver version: PHOENICS 2021 v1.0 Published by: Concentration Heat and Momentum Limited (CHAM) Confidentiality: Free Access

The copyright covers the exclusive rights to reproduction and distribution including reprints, photographic reproductions and translations. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, electrostatic, magnetic tape, mechanical, photocopying, recording or otherwise, without permission in writing from the copyright holder.

©Copyright Concentration, Heat and Momentum Limited 2023

1	Intro	oduction	2
2	Geo	metry	2
3	CFD	Analysis	2
	3.1	Creating the Domain	. 2
	3.2	Creating Fluid Boundaries	3
	3.3	Meshing	4
	3.4	Running the Simulation	6
	3.5	Results	6

CHAM, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK Telephone: 020 8947 7651 Fax: 020 8879 3497 E-mail: <u>rhinocfd@cham.co.uk</u>, <u>phoenics@cham.co.uk</u> Web site: <u>http://www.cham.co.uk</u>

1 Introduction

The tutorial describes a simple calculation of flow around a sphere and various methods of visualizing the results. It is assumed that the user is familiar with Rhino; if not, this <u>useful introduction</u> is helpful in getting you up and running.

2 Geometry

In a 'small objects – Meters' Rhino environment, create a sphere of any size using the sphere tool in Rhino.

3 CFD Analysis

3.1 Creating the Domain

Create the domain (fluid region) with the 'create domain to fit objects' button on the RhinoCFD toolbar:



A file find dialog will then appear; use this to locate a directory where you wish to work preferably an empty directory to make the process clearer, as all the RhinoCFD intermediate and result les will be saved here.

Specify the Working	Directory	-	-		_		×
G 🖉 🚽 « aaa	 duckyf 	eb22 🕨 FirstC	ase	▼ 49	Search FirstCas	е	م
Organize 🔻 Nev	v folder					•	
💱 Dropbox	^ N	ime	<u>^</u>		Date modified	Тур	5
Desktop Desktop Documents Music Pictures Videos Momegroup Geoff Michel	III		No items mate	ch your search			Select a file to preview.
🖳 Computer 📬 Network	+ 4				1	•	
	File <u>n</u> ame:	cad.3dm		•	All compatible fi	ile types	(*.*) ▼ ancel

Figure 2: Working Directory Selection

The next choice is the type of problem to be solved; choose Core.

Open case in	Choose menu option	
in current directory	Core	•
O in case directory	Domain to use polar grid	
OK	Cancel	

Figure 3: Type of Simulation Selection

Resize the domain by scaling or by using the Gumball tool - Type `Gumball On' if you cannot see the Gumball. Make the domain significantly larger than the sphere.



Figure 4: Domain and Sphere

3.2 Creating Fluid Boundaries

Now we need to set the inlet, outlet and other boundary conditions. In general, you must always add at least one inlet and outlet, or nothing interesting will happen. The easiest way to do this is via the Domain Settings. Right click the 'edit solution parameters' button in the RhinoCFD toolbar.

nain Settings	A 1.				? <mark></mark> ×
Geometry Sources	Models Numerics	Properties	Initialisation Output	Help	Top menu OK
Title of o	n Faces current Simula	tion			
RI	HINO-CFD MAIN	MENU Version	2017 dated 01/01	/17	

Figure 5: RhinoCFD Main Menu

Click on Domain Faces to add an inlet and outlet at low and high X locations of the domain. Click on settings and change the X velocity to 1m/s. Click OK and a reminder window will appear which you can also close by clicking OK.

Set Domain Edge	Conditions		-	-	-	-	— ×	
Choice: WALL - OPEN - FLOW - WIND - For syn	Choices for domain edge boundary conditions: WALL - impermeable friction boundary (PLATE) OPEN - fixed pressure boundary (OUTLET) FLOW - fixed flow boundary (INLET) WIND - Wind profile / fixed pressure For symmetry condition set all to No							
Xmin: Xmax:	Wall Wall	No	Open Open	No	Flow Flow	Yes	Settings	
Ymin:	Wall	No	Open	No	Flow	No	Settings	
Ymax:	Wall	No	Open	No	Flow	No	Settings	
Zmin:	Wall	No	Open	No	Flow	No	Settings	
Zmax:	Wall	No	Open	No	Flow	No	Settings	
WIND	No	Setti	ings					
			Canc	el		ОК		

Figure 6: Domain Face Options

Act as: Export		D Import No
Nett area rati	1.0	00000
Inlet density	is	Domain fluid
density set t	1.18	9000 kg/m^3
Method		Velocities
X Direction	1.000000	m/s
Y Direction	0.000000	m/s
Z Direction	0.000000	m/s
Inlet turbuler	nce:	Intensity
Turb. intensit	y 5.0	8 000000
TuBorn Common	de	

Figure 7: Edit Object Attributes

The CAD appearance will change to include two new objects (the inlet and outlet). The inlet and outlet properties can be modified ed by selecting the inlet or outlet and clicking the 'edit CFD properties' button in the RhinoCFD toolbar.



Figure 8: Sphere, Inlet and Outlet

3.3 Meshing

The next step is to ensure the mesh is adequate to resolve all the features in the simulation. Of prime concern is the sphere, which needs reasonable number of cells to de ne the curves. A secondary concern is to ensure the mesh "looks good", that is, that there aren't sharp differences in size between one cell to the next. More information on RhinoCFD

grids can be found in our Meshing User Guide. To modify the mesh, click on the fourth button on the toolbar

Co-ordinate system	Tin	ne dependence		
Cartesian		Steady		
Cut-cell method	PARSOL	Settings		
	X-Auto	Y-Auto	Z-Auto	
Domain size	31.21440	24.91956	22.67550	m
Number of cells	40	38	39	
Folerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	3	3	3	
Edit all regions in	X direction	Y direction	Z direction	

Figure 9: Geometry Dialog

RhinoCFD has an automeshing mechanism, so you will see a grid appear automatically. Its parameters can be modified manually by clicking on "X/Y/Z Direction".

Co-ordinate system	Tin	ne dependence		
Cartesian		Steady		
Cut-cell method	PARSOL	Settings		
	X-Auto	Y-Auto	Z-Auto	
Domain size	11.93447	11.98188	2.194563	m
Number of cells	60	57	38	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	14	13	5	
Edit all regions in	X direction	Y direction	Z direction	

Figure 10: Mesh Around Sphere

Alternatively, the mesh can also be modified simply by first displaying it (by clicking on the 5th button on the toolbar) and then double clicking on the mesh. This brings up a slider bar which can modify the mesh automatically in each direction.

Minimum cell fraction (%)	1.445440	
🔘 Minimum cell size (m)	0.272619	
Number of cells	40	
)	
● X direction ◎ Y direction	C Z direction	
● X direction ◎ Y direction ✓ Enable Auto meshing	C Z direction	

Figure 11: Automesh Dialog

To ensure this simulation runs quickly, use the slider or the mesh menu to get approximately 40 cells in each direction. (NOTE: If the slider is not allowing your mesh to go low enough, then enter the mesh menu enter any direction's menu and click on "set default").

3.4 Running the Simulation

Once the case set up is complete; run the solver with the 'run solver' icon in the RhinoCFD toolbar and select 'run' on the dialog that appears. The convergence monitor will then appear:

RhinoCFD August 2017 - EARTH- 64 bit Single Precision	na 2 konsta 2 k	a 1 kin 2 kin	ten (1 fan San 1	
Spot Values at (33, 21	, 20)	ş	Error - Cu	t off 1.000E-02 %
Value Change Low	High	Variable	Max	% Error Change
6.73E-01 -4.47E-03 0.00E+00	1.00E+00	Ul	1.00E+02	3.66E-01 -1.68E-02
2.90E-02 1.31E-03 -6.00E-02	3.00E-02	Vl	2.00E+01	1.98E-01 -9.41E-03
-3.06E-02 2.98E-04 -4.00E-02	0.00E+00	Wl	1.26E+01	1.61E-01 -9.29E-03
3.53E-03 2.45E-06 2.00E-03	9.00E-03	KE	6.31E+01	1.72E-01 -9.65E-03
	0.000.00	-	7 047.01	
2.39E-04 -2.93E-07 0.00E+00	2.00E-03	E.P	7.942+01	2.40E-01 -1.14E-02

Figure 12: Convergence Monitor

Your simulation will have finished and achieved convergence when your monitor displays similar results to the following:

Spo	ot Values a	t (12, 8,	6)	ę	Error - Cu	t off 1.00	0E-02 %
Value	Change	Low	High	Variable	Max	% Error	Change
-1.67E-03							-1.44E-04
1.01E+00	0.00E+00	0.00E+00	2.00E+00	U1	1.00E+02	8.69E-03	-9.44E-04
-1.35E-02	-7.17E-08	-2.00E-02	0.00E+00	Vl	1.58E+01	7.75E-03	-8.16E-04
-1.26E-02	-3.63E-08	-2.00E-02	0.00E+00	W1	1.26E+01	6.65E-03	-7.55E-04
2.37E-03	0.00E+00	2.00E-03	3.00E-03	KE	6.31E+01	3.60E-03	-3.67E-04
1.56E-05	0.00E+00	1.00E-05	2.00E-05	EP	1.00E+02	2.93E-03	-2.44E-04

Figure 13: Converged Solution

Please see the <u>User Manual</u> for more information on convergence.

3.5 Results

Load the results using the 'load results' icon in the RhinoCFD toolbar. A dialog asking for the result le name will appear.

File names	?
Files to be used for plotting: Use current result files Use intermediate sweep files User-set file names Latest dumped files	No No Yes (phida,pbcl.dat)
Cancel	OK

Figure 14: Load Results Options

Click OK to load the results from the latest dumped files. For a typical case you will create a range of contours to inspect the ow eld. The eld variable is initially set to P1 (Pressure). Figure 15 shows typical results:



Figure 15: Pressure Contour in Z Plane

Try moving and rotating the cutting plane using the Gumball to inspect different in the domain.



Figure 16: Pressure Contour and Shifted Cutting Plane

A selected cutting plane can be hidden by left clicking on 'hide current object (selected)' button in the RhinoCFD toolbar. Right clicking on the same button will bring all previously hidden objects back.

The viewer options menu/results panel can also be closed by clicking on 'OK'. It is then brought back by left clicking on 'edit display parameters'. The result properties menu contains all the settings to control selected cutting planes, streamlines, IsoSurface probes, line plots and contour plots. It is useful to note that if multiple probes are active in the Rhino environment, then any change to any tab of the results panel will change the settings of all probes, unless an individual plot probe has been selected first.

Viewer Options						
Cut Planes	Streamlines	Isosurfaces	Line probes	Surfa	ce Contours	
Appearance			Probe controls			
Colour map	Blue-yellow	v v	Locate Prob	be	New Probe	
Scale Min	 80	Reset	Scalar	Pressu	re	\sim
Scale Max	50	Reset	Plane style	Fi	illed	\sim
Colour bands	32		Minimum vector	size 0	.050000	
Averaged	Invert C	Colours	Maximum vector size		.000000	
Show maximum and minimum points Hide values outside of range		points	Vector Scale		.000000	
Opacity	•		Enable 3D ar	row		
Export option Export as	is Fext file	 ✓ Export 	Data source Case ph	i.case Load dat	ta from saved case	~ •
				Ok	Apply	

Figure 17: Cut Plane - Results Panel

The displayed variable can be changed from the drop down list at the top. There are also various styles of cutting planes that can be chosen, which can be used to display the results in a different way. Figure 18 shows the available options.

Plane a	as:
Filled	-
Filled	
Lines	
Vectors	
Tubes	
EdgedFill	
Vectors &	Fill

Figure 18: Plane Type Drop Down

Streamlines, IsoSurfaces, line probes and surface contours can be generated via the drop down 'Add probes from drop down' and choosing the desired probe. Click on the streamlines option to produce something similar to Figure 19. In this case the stream probe has been resized to stretch across the domain using the Gumball.



Figure 19: Streamlines

The streamline cylinder probe can be rotated to generate streams in other planes, as seen in the previous figure. The streamline properties can be viewed and edited by clicking on the stream tab on the results panel:

Viewer Options	;					
Cut Planes	Streamlines	Isosurfaces	Line probes	Surfac	ce Contours	5
Appearance			Probe controls			
Colour map	Blue-yellov	v v	Locate Prob	e	New Prob	e
Scale Min	-80	Reset	Scalar	Pressure	e	~
Scale Max	50	Reset	Streamline shape	2	Line	\sim
Colour bands	32		Frequency of ba	ll release	None	\sim
	Invert C	Colours	Number of strea	mlines	12	
Clip value	Clip values outside scalar key range		Streamline source shape			\sim
-						
Opacity						
Specular	• •					
Export option	IS		Data source			
Export as	Text file	 Export 	Case phi	i.case		\sim
				Load data	a from saved c	ase
				Ok	Арр	ly

Figure 20: Streamlines Results Panel

This panel allows the number of streamlines, the type of stream line and the source type to be changed, as well as the other the same variable and range options seen in the cut plane tab.

Change the number of stream lines to 50 by editing the 'No Streams' input box and typing 50. This will produce a greater number of streams across the stream probe. Try changing the 'Stream as' to tubes and lines to see a difference in display.

Select 'IsoSurface probe' from the RhinoCFD toolbar and a probe will appear in the domain. To change the IsoSurface options, click on the IsoSurface tab on the results panel.

Viewer Options	;				
Cut Planes	Streamlines	Isosurfaces	Line probes	Surface Contours	
Appearance			Probe controls		
Colour map	Blue-yellov	N V	Locate Prob	New Probe	
Scale Min	-80	Reset	Scalar	Pressure	~
Scale Max	50	Reset	Drawn on		~
Colour bands	32		IsoSurface value	e þ.000000	
	Invert C	Colours		Use probe value	
Opacity					
Specular	•				
Export option	15		Data source		
Export as	Text file	 Export 	Case phi	i.case	~
				Load data from saved case	
				Ok Apply	

Figure 21: IsoSurface Results Panel

On the third row, click in the tick box to use a user de ned value to colour and locate the IsoSurface. If this box is left ticked, RhinoCFD will use the value the 'mesh probe' is reading at its current location, and the IsoSurface value is

changed by moving the probe. In the box labelled IsoSurface, edit the value to be in between the max and min values of the selected variable. The IsoSurface will then change to show regions of the specified value. Figure 22 shows typical results.

Figure 22: IsoSurface

To create a surface contour, click on the 'select and contour blockage' in the probe drop down list and then select the sphere. This should produce something similar to the following figure:

Figure 23: Surface Contour

The transparency of the surface contour can be changed by choosing an option from the 'Opacity' list. The data used to colour the surface can be exported as a .csv le, by clicking on 'export results' and selecting the desired le location.

at i falles	Streamlines	Isosurfaces	Line probes	Surfa	ce Conto	ours
Appearance			Probe controls			
Colour map	Blue-yellow	w ~	Clear Surface C	Colours	Contour	an object
Scale Min	-80	Reset	Scalar	Pressur	re	
Scale Max	50	Reset				
	32					
Colour bands	32					
Colour bands	Invert (Colours				
Colour bands	Invert (Colours				
Colour bands	Invert (Colours				
Colour bands	Invert (Colours				
Colour bands	Invert C	Colours				
Opacity	Invert (Colours				
Opacity Specular Export optior	Invert (Colours	Data source			
Opacity Specular Export option Export as	Invert (Colours	Data source Case pr	i.case		

Figure 24: Surface Contour Results Panel

The data at any point, or along a string of points, in the domain can be exported as a .csv le using the line probe.

Viewer Options	;					
Cut Planes	Streamlines	Isosurfaces	Line probes	Surfa	ce Contours	
Appearance Colour map	Appearance Colour map Blue-yellow ~			e	New Probe	
Scale Min	-80	Reset	Scalar	Pressur	e	~
Scale Max Colour bands	50 32 Invert C kimum and minimum	Colours	Number of points on line	þ 6		
Export option Export as	ns Text file	 ✓ Export 	Data source Case ph	i.case Load dat	a from saved case	~
				Ok	Apply	

Figure 25: Line Results Panel

Using the 'line' tab of the results panel click on 'Add Probe' and then move the generated probe around the domain:

Figure 26: Line Probe

Then enter 100 as the number of 'points on line graph' and select 'Export results'. This can then be opened and plotted in a program such as Excel.

It is possible to display multiple cutting planes, iso-surfaces and stream lines at the same time. To create a new probe either click on the desired probe button from the RhinoCFD toolbar or 'create probe' from the results panel; or to duplicate a probe, press and hold down Alt and then drag the desired probe using the Gumball.

Figure 27: A Range of Cutting Planes and Probes

To remove the visualizations and return to the Pre-processor stage, right click on 'load results'. From here you can change any settings and run the simulation again.