

PHOENICS Case Study: HVAC Air Flow and Solar Radiation

PHOENICS/FLAIR, with a structured computational mesh, was employed, by the T C Chan Centre for Building Simulation and Energy Studies in Pennsylvania, USA, to create the following replication of a pseudo-3D tutorial example for the built environment.



Modelling goals:

- 1. Outdoor wind flow into indoor space.
- 2. Heat source to change the inlet temperature of wind as application of modern wind tower.
- 3. Solar radiation outside as another heat source from natural system \rightarrow Wind Speed and Distribution of heat to indoor and surrounding.

Although it is straightforward to set up the geometry of such a case using PHOENICS 'primitives', in this scenario the building design in question has been imported from CAD as a DWG format file.

The facility exists within PHOENICS/FLAIR to read weather data prevailing for any particular location from an external data file such as EnergyPlus. In this example, the wind direction and solar gain values are individually specified via the menu. The 2D representation of the building in question is as follows:

Concentration Heat and Momentum Limited, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK Telephone: 020 8947 7651 Fax: 020 8879 3497 Fax: 020 8879 3497 E-mail: phoenics@cham.co.uk, Web site: http://www.cham.co.uk





Starting a new model in PHOENICS

In the VR-Editor, go to File > New case. Select FLAIR as the interface best suited for this type of application/simulation.

Pre-processing

Set the model domain size through the main menu > geometry. This can be done as indicated below.

Settings View Run Onti-	ons Compile Build Help				
ettings view Kun Optio	▼ DOMAIN		4 t 5 % 7 R -		
	· · · · · · · · · · · · · · · · · · ·				
			MAIN		VR Editor
		L			
			IVIEINU		
ſ					
					Menu Ob
					<u>- #</u> 🗵
					Position X 0.500000
					Y 0.500000
					Z 0.500000
					Size
					× 30.00000
					Y 10.00000
					Z 12.00000
7					
<u></u>					
	▼				
Y L	— X				
	No 1111 - 1	at for this			
AIR	No title has been s	et for this run.			
AIR	No title has been s	et for this run.			
AIR	No title has been s	et for this run.			
AIR	No title has been so ind Mesh Settings	et for this run.			8 ×
AIR	No title has been so and Mesh Settings	et for this run.	_	-	<u>8 - 2 -)</u>
AIR	No title has been so frid Mesh Settings Co-ordinate system	et for this run. Tir	ne dependence		9
AIR	No title has been so irid Mesh Settings Co-ordinate system Cartesian	et for this run.	ne dependence Steady		2 2
AIR	No title has been so ind Mesh Settings Co-ordinate system Cartesian	et for this run.	ne dependence Steady		
AIR	No title has been so brid Mesh Settings Co-ordinate system Cartesian Partial solids treatme	et for this run.	ne dependence Steady Set	tings	
AIR	No title has been so wid Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size	et for this run.	me dependence Steady 	tings Z-Manual	
AIR	No title has been so wid Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size	et for this run. Timent On X-Manual 30.00000	ne dependence Steady Y-Manual 10.00000	tings Z-Manual 12.00000	9 <u>×</u>
AIR	No title has been so mid Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance	et for this run. Timent On X-Manual 30.00000 151	ne dependence Steady Set Y-Manual 10.00000	tings Z-Manual 12.00000 76 1.000 T6	m
AIR	No title has been so wid Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions	et for this run. Timent On X-Manual 30.00000 151 1.000E-3	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3	tings Z-Manual 12.00000 76 1.000E-3	m
AIR	No title has been so wid Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region	et for this run. Tim ant On X-Manual 30.00000 151 1.000E-3 5	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1	tings Z-Manual 12.00000 76 1.000E-3 4	m
AIR	No title has been so ind Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region Size	t for this run.	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1 10 10.00000	tings <u>Z-Manual</u> 12.00000 76 1.000E-3 4 1 2.0500002	m
AIR	No title has been so bind Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region Size Distribution	t for this run.	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1 10.000 Bower back	tings Z-Manual 12.00000 76 1.000E-3 4 1 3.050000 Power 1	m
AIR	No title has been so bind Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region Size Distribution Cell power	t for this run. Tim ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Sot	ne dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1 10.0000 Power law	tings Z-Manual 12.00000 76 1.000E-3 4 1 3.050000 Power law Sat	m
AIR	No title has been so ind Mesh Setting: Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region Size Distribution Cell _power Cells in region	t for this run. Tim ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Set 20	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1 1 0.00000 Power law Free	tings Z-Manual 12.00000 76 1.000E-3 4 1 3.050000 Power law Set 25	m
AIR	No title has been so and Mesh Settings Co-ordinate system Cartesian Partial solids treatmon Domain size Number of cells Tolerance No of regions Modify region Size Distribution Cell _power Cells in region Power/ratio	et for this run. Tir ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Set 20 -1.300000	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1 1.000E-3 1 1 1.000E-3 Free 1 1.000202	tings Z-Manual 12.00000 76 1.000E-3 4 1 3.050000 Power law Set 25 1.000000	m
AIR	No title has been so and Mesh Settings Co-ordinate system Cartesian Partial solids treatmon Partial solids treatmon Domain size Number of cells Tolerance No of regions Modify region Size Distribution Cell _power Cells in region Power/ratio Symmetric	t for this run. Tir ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Set 20 -1.300000 No	me dependence Steady Y-Manual 10.00000 1 1.000E-3 1 1 0.00000 Power law Free 1 1.00000	tings Z-Manual 12.00000 76 1.000E-3 4 1 3.050000 Fower law Set 25 1.000000	m
AIR	No title has been so mid Mesh Settings Co-ordinate system Cartesian Partial solids treatmon Partial solids treatmon	t for this run. Tir ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Set 20 -1.30000 No X direction	me dependence Steady Set Y-Manual 10.00000 1 1.000E-3 1.000E-3 1.00E-3	tings Z-Manual 12.00000 76 1.000E-3 4 3.050000 Power law Set 25 1.000000 No Z direction	m
AIR	No title has been and mid Mesh Settings Co-ordinate system Cartesian Partial solids treatme Domain size Number of cells Tolerance No of regions Modify region Size Distribution Cell _power Cells in region Size Distribution Cell _power Cells in region Symmetric Symmetric	et for this run. Tir ant On X-Manual 30.00000 151 1.000E-3 5 1 6.456749 Power law Set 20 -1.300000 No X direction	me dependence Steady Y-Manual 10.00000 1 1.000E-3 1 10.00000 Power law Free 1 1.00000 No Y direction	tings Z-Manual 12.00000 76 1.000E-3 4 3.050000 Power law Set 25 1.000000 No Z directior	m m n a

Next, all relevant objects are added to the domain. Clicking *Obj* on the control panel selects the following.

ect Action Vi	ew Group)				
New	•	New Object	by .	Visibility	Wireframe	
Copy object(s)		Import CAD Object		on	on	
Array object(s)		Import CAD Group		on	on	
Anay object(3)		Import CAD Group		on	on	
Select All		Import Object		on	on	
Clear All		Clipping plane		on	on	
Defeat		Diatting Surface		on	on	
Kerresh		Plotting surface		on	on	
Close		BLOCKAGE cub	pe14	on	on	

Select *New Object* and set object type to *Sun* object. Then, by clicking attributes, the following panel appears, for which relevant data can be specified.

Sun Attributes	ହ <mark>×</mark>
Get North and Up from	No
Use weather data file	No
Direct Solar radiation	From solar altitude
Date (dd/mm/yy) 21 Jun	
Time (24hr) 1:	1 h 0 m 0 s
Optional extra output	
	Cancel OK

Once this is done, again go to *Obj* > *New*, select *Import CAD Object* from the list and select the desired CAD file.

🛞 Object Manageme	nt					x
Object Action Vie	w Grou	р				
New	•	New Object	ry	Visibility	Wireframe	Affe
Copy object(s)		Import CAD Object		on	on	ууу
Array object(s)		Import CAD Group				
Select All		Import Object				
Clear All		Clipping plane				
Refresh		Plotting Surface				
Close	Ĭ					
						•
To create new object, s	elect Nev	v object from Object menu				
To create new object, s	elect Nev	v object from Object menu				1

Once it is selected, the following screen will appear. We have the option to scale the CAD object in question, which can be useful if units used for the CAD file are not in metres (PHOENICS default is to use SI units.)

Geometry Import	ि <mark>२२</mark>
Read 130 points and 404 facets	
The original size and position are:	
dx 1.585360E+00 dy 6.717100E-01 dz	1.00000E+00
x0 6.456750E-01 y0 0.000000E+00 z0	0.00000E+00
Take size from Geometry file	Yes
Take position from Geometry file	No
Geometry scaling factor 1.000000	Apply
CAD X Y Z align with VR X Y Z	
Object constrained by domain Ye	s
Cancel	

Once imported, clicking on attributes displays the following screen: The desired material can be selected, along with its roughness and its solar absorption factor.

		121 Concrete bl	ock (mediumwe	eight)	
Types:	Soli	lds			
Roughness		Default	0.00000	m	
Wall function	law	Default			
Slide Velocity	:	Stationary			
Energy Source:		Adiabatic			
Heat transfer (coeff	Wall function			
Solar absorpti	on 0.	200000			
	ature	No			
Initial Temper					

From the same *Obj* menu we can then add the remaining objects. First we will add an inlet. Select *New object* and from the drop-down list and select *Inlet*.

			0 8
Inlet Attributes			
Act as: Export	N	o Import No	
Nett area rati	• 1.	000000	
Inlet density	is	Domain fluid	
at an Ambie	nt pressur	e of: 0.000000	Pa
relative to	1.0	13E+05 Pa	
Temperature	Ambient	27.50000 °C	
	,		
Method		Velocities	
X Direction	3.00000	m/s	
Y Direction	0.00000	m/s	
Z Direction	0.00000	m/s	
Inlet turbulen	ce:	Intensity	
Turb. intensit	y 5.	000000 %	
InForm Command	ls		
]		
		OK	

Clicking attributes we can modify the inlet conditions.

Size and place are then modified and set to fill the entire left-side of the domain (using "to end" option), with an object position of (0, 0, 0).

Object Specification	8 X
General Options Size	Place Shape
Object position, m × 0.000000 ÷	Rot'n angle about axis, deg ☐ at end X 0.000000 → ☐ at end X 0.000000 →
Z 0.000000	□ at end Z 0.000000 ÷
ОК	Cancel Reset Apply

Likewise, an *Opening* (or outlet) object must be added on the far right of the domain. This is a pressure boundary, with flow through it governed by the difference in pressure inside the domain and the external reference pressure (normally set to ambient pressure). The position of this outlet is set to X "at end" or at 30m.

ning Attributes			8
Act as: Export	h No Imp	ort No	
Nett area rat:	io 1.000000		
External Ambi	lent pressure	0.000000 Pa	
Relative to	1.013E+05 P	a	
Coefficient	100.0000	Linear	
Text <u>Ambier</u>	ulence is:	°C	1
Velocity X	User-set	0.00000	m/s
Velocity Y	User-set	0.00000	m/s
Velocity Z	User-set	0.00000	m/s
	ds		
InForm Comman			
InForm Comman		OK	

Object Specification		8 ×
General Options Size	Place Sha	pe
_ Object position, m		Rot'n angle about axis, deg —
× 30.00000 ÷	🔽 at end	× 0.000000 ÷
Y 0.000000 ÷	🗆 at end	Y 0.000000 ÷
z 0.000000 ÷	🗖 at end	Z 0.000000 +
ОК	Cancel	Reset Apply

[Note that if this were a 3D simulation, it would be easier to use the *Wind* object to define the external boundary conditions through a single user panel.]

	NO				
ernal density is:	Domain fl	uid			
ernal pressure	101325.0	Pa			
fficient	1000.000	Linea	r		
ernal Temperature	27.50000	°C			
d speed	10.00000	m/s			
d direction	North	0.00000	•		
erence height	10.00000	m			
le between North and Y	¥ 0.000000	•			
file Type	Logarithmi	-			
tical direction	z				
ective roughness heigh	ht				
Ope	n sea			2.000E-4	m
lude open sky N	•				
lude ground plane N	•				
	Factor (WAN	4P)		No	1
Ope lude open sky N lude ground plane N	n sea o o Factor (WAN	4P)		2.000E-	-4

To model the cooling effect of the chimney due to a water spray, a blockage of *Domain* material type is added. This will overwrite any solid regions which happen to overlap, and in this case implies the thin vertical plates within the CAD geometry. Clicking *Attributes* we can change the material of the object to *Domain Material*, which in this case is air.

A *Linear Heat Source* source is added with a value (or temperature) set to 15°C, and a coefficient, which establishes the extent that this temperature is to be enforced, ie a large value of 1.0+E07 would ensure air passing through the patch is fixed at 15°, while a lower value such as 100 will allow more variation, dependent on the coefficient multiplied by the difference between the specified value and local temperature of the air.

ockage Attributes			ହ <mark>-</mark> ×
Material: 2	Air using Ideal Gas Law,	STP	
Types: Dom	ain Material		
-		1	
Energy Source:	Linear Heat Source		
Coefficient	100.0000 Value	15.00000	
Other Sources:	Momentum Source Initial Values		
	Porosities		
	InForm Commands		
	OK		

In addition to this a momentum source is set to restrict flow to the Z direction only and apply a small sink using a *Quadratic Source* to provide a pressure loss based on velocity squared via an appropriate *coefficient*.

			0 00
X direction:	Fixed	Velocity	
Value	0.00000	m/s	
Y direction:	Fixed	Velocity	
Value	0.00000	m/s	
Z direction:	Quadrat	ic Source	
Coefficient	1.000000	Value	0.000000
	[OK	

One more object is necessary for this model, and that is the Ground terrain. It is added in the same fashion as the objects above, selecting *Plate* as the type of object, sizing it to cover the entire XY plane and located at (0,0,0). This object will provide the no-slip condition for the ground, and boundary layer development for a smooth surface.

object Action	<u>i icii oroup</u>	1-	1 -	1		-
Object name	Reference	Туре	Geometry	Visibility	Wireframe	Affe
DOMAIN	-1	DOMAIN		on	on	ууу
MERGED_1	0	BLOCKAGE	mygeom_3	on	off	ууу
IN	1	INLET	cube3t	off	off	ууу
OUT	2	OPENING	cube12t	on	off	ууу
SUN5	3	SUN	sun	off	off	ууу
GROUND	4	PLATE	cube11	on	off	ууу
CHIMNEY	5	BLOCKAGE	cubet1	on	off	ууу
•		m				+

With the geomety set up, the domain settings are applied.

Clicking the *Menu* button on the control panel, and navigating to *Models* allows the turbulence model to be selected, and ensures the energy equation is solved (*FLAIR* default settings are fine).

n Settings	2 <mark>- ×</mark>
Geometry Models Properties	Initialisation Help Top menu
Sources Numerics	Output
Equation formulation	Elliptic-Staggered
Lagrangian Particle Tracker (GENTRA)	OFF
Solution for velocities and pressure	ON
Energy Equation	TEMPERATURE TOTAL
Turbulence models	KECHEN settings
Radiation models	OFF
Fan operating point	OFF
System Curve	OFF
Comfort indices	settings
Solve Specific humidity	OFF
Solve smoke mass fraction	OFF
Solution control / Extra variables InForm - Group 7	settings Edit InForm 7
More Page Dn Line Dn	

The domain material is set to *Air using Ideal Gas Law*, which allows for temperature and density variations and ambient conditions to be set.

ain Settings	<u>୧</u> ×)
Geometry Models Properties Sources Numerics	Initialisation Help Top menu Output
Domain material:	
The current domain material is	2 Air using Ideal Gas Law, STP
Edit properties of current mater.	Temperature units Centigrade
Ambient pressure 0	Ambient temperature 27.5 C
Initialise from ambient ON	Set buoyancy from ambient ON
Initialise from ambient <u>on</u> Property storage	Set buoyancy from ambient ON
Initialise from ambient <u>ON</u> Property storage Prandtl/Schmidt Nos settings	Set buoyancy from ambient ON
Initialise from ambient ON Property storage Prandtl/Schmidt Nos settings	Set buoyancy from ambient <u>ON</u> InForm - Group 9 <u>Edit InForm 9</u>

Buoyancy is set in the *Sources* panel. This option is set automatically to the *Density Difference* method if *FLAIR* is used.

The option to have buoyancy affect turbulence can also be set in this panel (eg for fire/smoke modelling).

Sources	Numerics	Properti	es In	Output	Help	Top menu
Gravitationa	al forces	ON				
Buoyancy mod	del is	DENSITY_DI	FFERENCE			
Gravitation	al 0.0000	¢ 00	¥ .000000	Z	000	
Reference de	ensity (kg/m	^3) 1.1	74864	(Deduced fr	om Ambient	conditions
Buoyancy ef:	fect on turb	ulence	off			
Buoyancy ef: Cyclic bound	fect on turb dary conditio	ulence	OFF	•		
Buoyancy ef: Cyclic bound Coeff. for : Global wall	fect on turb dary condition auto wall fun roughness	ulence ons <u>ALL_s</u> nctions	OFF	DG-LAW		

Solver conditions are set under *Numerics*. These include the total number of iterations to be performed, the type of differencing scheme to be used and the settings of any relaxation parameters.

n Settings		8
Geometry	Models Properties Initialisation Help Top n Numerics Output	nen
Total nu Minimum	mber of iterations 4000 number of iterations 1	
Maximum	runtime Unlimited	
Global (convergence criterion 0.010000 %	
Relaxa	tion control Iteration control	
Limits	on Variables Differencing Schemes	
	Multi-Grid Accelerator - MIGAL	
Converge	Edit InForm Group <u>15</u> <u>16</u> <u>17</u> <u>18</u> ance accelerator for highly compressible flow:	
More	Page Dn Line Dn	

The default option for relaxation is *Automatic Convergence Control* (CONWIZ): On. This is a convergence wizard that automatically sets the relaxation parameters to try to ensure convergence. If it is switched off, experienced users can set parameters manually, which can lead to faster run times using less conservative relaxation values.

Relax	ati	Pr	evious pan			
Automati	c C	onvergence (Control _	ON	Reset solu	tion defaul
Variable	>	P1	U1	W1	KE	EP
RELAX		LINEAR	FALSDT	FALSDT	LINEAR	LINEAR
VALUE		1.000000	0.100000	0.100000	0.500000	0.500000
MAXINC		1.000E20	100.0000	100.0000	0.100000	1000.000
Reference v	elc	city 10	.00000	Reference le	ength 1.000	0000

PHOENICS creates automatically a Cartesian mesh across the whole domain. It uses a cut-cell algorithm called *PARSOL* (<u>PARtial SOL</u>ids) to analyse which parts of a given cell are occupied by fluid and solid regions. This reduces greatly the complexity of generating a mesh for any given case.



Clicking the mesh button on the control panel displays the mesh overlaid on the domain. Each of the regions (created by objects affecting the grid and denoted by orange lines on the mesh display) can be modified individually by clicking them and changing the desired parameter on the *Grid Mesh Settings* panel.



In certain regions it might be beneficial to crush the mesh using non-uniform spacing to reduce the overall cell count. Both *Power* and *Geometric Progression* distributions are available, with power/ratio settings <-1 for diminishing and >1 for increasing mesh spacing.

Once the desired mesh is obtained, the solver can be run. To commence the run, select *Solver* from the *Run* menu tab.

Co-ordinate system	Tir	ne dependence		
Cartesian	_	Steady		
Partial solids treatme	ent On	Set	tings	
	X-Manual	Y-Manual	Z-Manual	
Domain size	30.00000	10.00000		
Number of cells	151	1	76	
Tolerance	1.000E-3	1.000E-3	1.000E-3	
No of regions	5	1	4	
Modify region	2	0	4	
Size	8.863251	0.000000	5.282900	
Distribution	Power law	Power law	Power law	
Cell _power	Set	Free	Set	
Cells in region	59	0	20	
Power/ratio	1.000000	0.00000	1.300000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	

A real-time plot of residuals and spot values for the probe location are printed to the screen as the run progresses (known as a GXMONI plot). This establishes whether the solution is converging or not. As we can see on the right hand side of the plot below, the residuals tend to be very small numbers, while the spot values on the left remain stable as the number of iterations increases - an indication of good convergence.



Post-processing

Once the run has completed with satisfactory convergence, select *GUI*, from the *Post-Processor* tab (also under the *Run* menu tab). Clicking the *C* button on the control panel opens the contour plot options.



Vectors can be used in conjunction with, or instead of contour plots by selecting them from the Vector tab.

Use of the Sun object provides the option to plot certain related field variables such as #Sol (Solar absorption factor), #QS2 (Total heat source per unit cell) or LIT (Illumination flag).





These can be accessed from the *Current Variable* drop-down menu.

Full post-processing guidance can be found by clicking on the PHOENICS-VR Reference Guide: <u>http://www.cham.co.uk/phoenics/d_polis/d_docs/tr326/tr326top.htm</u>