

1D user-manual

INTERNATIONAL CENTER FOR NUMERICAL METHODS IN ENGINEERING

Gran Capitán, s/n 08034 Barcelona, Spain

Publications CIMNE N°PXXX - , July 2012

International Center for Numerical Methods in Engneering

Edificio C1, Campus Norte UPC

Gran Capitán, s/n

08034 Barcelona, Spain

www.cimne.upc.es

First Edition: Junio 2012-07-16

1D User MANUAL FOR CARDIOVASCULAR PROBLEM

IT- CIMNE

© The Authors

Summary

Introduction	5
Model reproduction	7
Problem Data	9
Boundary conditions	12
Interval Data	16
Mesh generation	17
Running test	19
Results analysis	20

Introduction

Using GiD pre-process tool is possible to design a model of the whole artery system of human body. GiD is very user-friendly and more in the case that the user wants to work only in 1D condition. The draw the arteries is it possible to use the automatic creation tool for lines or points. This tool does possible to create the arteries of the vessel system or other test vessels with the right



It is useful to remember that it is not necessary that the model designed represents the whole artery system of the human body; it is, of course, possible to reproduce only a part of the entire system, like the legs or an arm, or reproduce any kind type of 1D system that the user wants.

When the user has drawn the model, reproducing the characteristics and the dimensions of the real artery system, like showed in figure 1, and the process of modelization is finished the user has to charge the problem type, in order to upload all the conditions and the tools thought to solve this kind of problems. To upload the problem type configuration, chose from the principal tool bar the option

Data > Problem Type > ARTERIES_1D

like is showed in figure 3. Remember that before of this procedure the user must to install the problem type inside GiD program.



Model reproduction

For a correct reproduction of the system, it is not important to reproduce the position or the inclination of the vessel. User is working in 1D environment. The position of the vessel is only an help for the user, in order to have a clear vision of the problem. Arteries 1D does not mind if 2 o more arteries are close, the relationship between the vessels will be assigned to the user, specifying the number of the vessel related and the type of connection between them.

In order to start designing the model is important to do an overview of the GiD tools for the geometry CAD utilities. The basic item for the models of problem type Arteries 1D is the line. From the GiD tool bar choose:

Geometry > Create > Straight line

In the screen the user could chose the first and second point of the line or, if the user want, will be possible to introduce manually the coordinates of the two points. This operation will be repeated every time that the user wants to create a line, every line represents an artery.



The real important thing that the user has to reproduce is the dimension of the vessels. 1D Arteries takes and considers for the FEM calculation the dimensions that the user has drawn.

It is very important to highlight that the normal of every line created may be concordant with the flow direction, otherwise the program will not be able to reproduce the right evolution of the flow, inside the model. Some examples are showed in the next pictures.



Problem Data

First of all it is necessary to define the general conditions of the problem that the user wants to perform. In the window represented in figure 3, it is possible to define the physical properties of the blood (or maybe other kind of fluid, if the user wants), specifying the blood density and the dynamic viscosity. User also must define the geometrical profile of the imposed velocity, the number and the duration of the cardiac pulse and finally the velocity (or pressure or flow) profile that will be imposed at the inlet. From the GiD tool bar select

Data > Problem Data

The window showed in the next picture will appear. In this window it will be possible to set several general conditions of the problem:

Title: is the name of the file that the program will generate. By default the program set this field like "Default_title". It means that the entire related file generated will have the same name of the project in which we are working.

Blood density: is the density of the fluid simulated. The default value is the most common value of blood density found in medical literature.

Blood dynamic viscosity: is the viscosity of the fluid used in the program. May attention, please, to the units of this value. It is a **dynamic** viscosity expressed in [kg]/[m][s].

Aortic Initial Pressure: is the basic pressure of the system.

Constant hydrostatic pressure: is the external pressure applied to all the model

Geometrical Flow Profile: this field gives the possibility to use a default profile of flow (or velocity or pressure). This field will be ignored if the user puts some values in the field **Inlet Profile Data**.

Number of cardiac cycles: indicates how many times the inlet profile will be repeated in the test.

Inlet Profile Data: in this field the user can configure the profile of the flow in the inlet section. The data have to be inserted in the grid, has the structure showed in table 1. Note that it is possible to copy the data directly from external text files and paste them in the grid. It is also possible to plot the data inserted, in order to verify the correctness of the profile wanted.

0.00	0.303554897
0.01	0.322612797
0.02	0.348995441
0.03	0.381120324
0.04	0.418972373

Table 1 Example of velocity profile. The data are imposed in the selected inlet point.

Problem Data				
GENERAL OUTPUT VIEW				
Title Default title				
	Κα			
Blood density 1021	m ³			
Blood dynamic viscosity 0.004	Kg m·s			
Aortic Initial Pressure 80	mmHg			
Constant hydrostatic pressure 3.2	mmHg			
Geometrical Flow Profile[s(v)] Parabolic	•			
Number of cardiac cycles 1				
Inlet Profile Date	acl Magnitude [m/l/cec]	•		
Inter Prome Data Interes	magnitude [m///sec]			
Accep	t <u>C</u> lose			
Problem Data		8		Using this field user can set the
		2		inlyt profile. A monu will appear
GENERAL OUTPUT VIEW				inter prome. A menu wiii appear,
The Default_the	_ Kq			and the user can write or paste the
Blood density 1021	m ³			
	Kg			value of the profile.
Blood dynamic viscosity 0.004	m·s			
Aortic Initial Pressure 80	mmHg			
Constant hydrostatic pressure 3.2	mmHg			It is also possible to draw the
Geometrical Flow Profile[s(y)] Parabolic	-			
Number of cardiac cycles 1	10			inserted profile using the button
Inlet Profile Data Time [sec] Magnitude [m/l/sec]			
0.00	0.303554897	^		on the right
0.01	0.322612797			
0.02	0.381120324			
0.04	0.418972373		J	
0.05	0.458813859	_		
0.06	0.500132351	=		
0.07	0.582151405			
0.09	0.620892672			
0.10	0.656928606			
0.11	0.689852278			
0.12	0.719264292			
0.13	0.741336491			r
0.14	0.757282128			
0.16	0.770341575			
0.17	0.765119303		7	
0.18	0.755006333			
0.19	0.741389241			
0.20	0./25443604			
0.21	0.69103539			
0.23	0.673808676			
0.24	0.655768101			
0.25	0.636868452	-		
î				
	Figure. 5 Window for t	the selection of th	ne tvpe	e of inlet profile
				· · · · · · · · · · · · · · · · · ·

Problem Data				
GENERAL OUTPUT VIEW				
Vrite GiD Results				
Write Text Results				
Time step to share 0.01 S				
Show section flow in postprocess				
Show section pressure in postprocess				
Show section velocity in postprocess				
Show section transversal displacements in postprocess				
Results format file Binary 🔻				
<u>Accept</u> <u>Close</u>				
Figure. 6 Window for the selection of the type of inlet profile				

The second window of the menu **Problem Data** gives the possibility to select how results write in the output files, and through the field

Section contour nodes: is the quality of the render for the 3D visualization of the cilinders that represent the vessels

Number of step to share: how many results write in the output results file.

The other selections permit to choose how visualize in the post process files.

Finally we set the output file format: Binary format is the default option.

Boundary conditions

Once the user finishes drawing the model, he can start to set the conditions for the computation. First of all, it is important to set the *Inlet* condition, specifying the kind of inlet that will be imposed in the model. User can choose between *flow*, *velocity* or *pressure*, like shown in figure 3.



The second step is the assignation of the peripheral resistances. This parameter is extremely important because is the responsible of the pulse wave or rather than the pressure reflexion. That wave can increase the pulse curve profile, if the principal wave and the reflected one are added together or can reduce the peaks of the wave if the waves are in phase contraposition. In order to obtain the right pressure response, is extremely important that the user introduce the right value that can change for any type of simulation or case of study.



In humane cardiovascular system there are a lot of bifurcations, and in some cases the scientist may occur in a confluence between two arteries in only one vessel. In other cases, when for example there is an obstruction in the vessel or an aneurysm may be interesting to reproduce the change of section or the properties of the vessel. *Arteries 1D* permits to reproduce and simulate these kinds of connections. User has to select how neighboring arteries are connected to each other. It is necessary to select if the relationship between 2 or 3 arteries is a bifurcation, a confluence or the connection is a simple change of properties in the same vessel. (figure 5).



An important feature to emphasize, for the right assignation of the bifurcation condition, is that is not allowed to assign the same bifurcation number at two connection of the same type. Instead it is possible to assign the same number of bifurcation if the bifurcation are different (for example the first one is 2-1 and the second is 1-2).

If the user makes confusion and assign the same bifurcation number too many different connections, the program understand that entire node are joined but in a not allowed configuration. User are also obliged to decide and set how is the node (or nodes) effluent and how is the node (or nodes) affluent. To do this, the user may use the windows shown if figure 6.



It is also possible to control visually all the conditions (boundary or connections) GiD permit to draw color for each condition, separately or not (figure 7). This is a very smart tool due to the complexity of the models that we normally use.



Every artery needs to have a few properties like: length, radius, thickness, Young's module and Poisson module and the mean wave speed.

For this reason *Arteries 1D* owns a library with the characteristics of the 63 principal arteries in the human body. User can also modify these values to perform simulation of pathological cases or to reproduce some unusual situation. *Network blood vessel* window (figure 8) will permit to set those properties.

	Network blood vessels	
C	1 Ascending aorta 🔹 🗸 🏹 🗸	
	radius 0.0147 m	
	thickness 0.00163 m	
	E 0.4000E+06 Pa	For any vessel that the user reproduces
	Poisson module 0.5000	Tor any vesser that the user reproduces
		it is obligatory to assign the kind of
		vessel that it is. Every vessel needs a
	<u>A</u> ssign ▼ <u>D</u> raw ▼ <u>U</u> nassign ▼ Exchange	different and unique name.
		The fields are filled by default with the
	Network blood vessels	most common values founded in
<	6 R. vertebral 🔹 😨 🔀 🗶 🖉 🗸	literature.
	radius 0.0024 m	
	thickness 0.4500E-03 m	It is possible to modify every value if
	E 0.8000E+06 Pa	the user wants it.
		Finally the properties will be assigned
	Assign • <u>Daw • Unassign</u> Exchange	Thiany the properties will be assigned
	Close	to the LINE that represents the vessel.
	Figure. 12 Network blood vessels windows. This menu gives the po reproduced	ssibility to assign or modify the properties of the arteries I.

Interval Data

The last data to set are the interval data. This menu offers to the user the possibility to set some technical and theoretical parameters. As for the previous menus some values and options are charged for default and in many cases are considered like the best choose. At any rate, in order to give more freedom to the user, it is possible to modify those parameters as well as the solver or the integration method used by the program.

1		$\overline{}$ \otimes \times	2
FEM solver	Taylor Galerkin		
Time-integration method	4th order Runge-Kutta		
Max steps number	40000		
CFL [step limitation parameter]	0.5774		
Sigma [Correction parameter]	0.125		
✓ Use Adaptive Stepsize Cont	ol		
Min Step	1e-07 s		
Li tolerance	1e-03		
Newton-Rankson tolerance	1e-05		
Newton-Kaphson tolerance	16-00		
	Accept	Close	

The fields that the user must decide are:

FEM solver: the formulation used to solve the problem.

Time-integration method: is the method selected for the integration in the time

Max step number: establish the maximum number of operation that the program can do to solve iteration.

CFL and Sigma: are parameters needed to stabilize the computation, CFL restricts the possible size of time step, SIGMA is used to correct the solution.

MIN step is used as inferior threshold for the size of the time step, is considered the minimum value for time step calculation.

U tolerance and Newton-Rapson tolerance: are the parameters that set the tolerance for the error in the computation of the solution.

Mesh generation

When the model is completed and every condition, parameters or values are stabilized and assigned the user must generate the mesh.

Selecting from the tool bar the option

Mesh > Generate Mesh

Project: Full_S	SPA3 (Arteries_1D)	
eometry Utilities Da	ata Mesh Calculate Help	
ୖୖୖୖୖ୕⊗ୖୖୄୖଢ଼ୄ୕ୖ୕ୄ୴୲୶	Unstructured Image: Constructured SemiStructured Image: Constructured Cartesian Image: Constructured	
	Boundary layer	
	Element type	
	Show errors Ctrl-g	
	View mesh boundary Create boundary mesh	
	Mesh guality	
	Mesh options from model	
	Figure. 14 Generating a mesh with GiD.	

Or with the combination of the key *Ctrl+G*, GiD will open the follow window:

	Mesh generation	
	Enter size of elements to be generated	
	Get meshing parameters from model	
	<u>Q</u> K <u>C</u> ancel	
Figure, 15 Mesh generation parameter window.		

Where the user will decide the maximum value of the element that the mesh generator will create. If the model was correctly created a warning window will appear, it will show the quality of the created mesh: how many elements and nodes it has generated, like showed in figure 16:

Dialog window	3
Mesh generated.	
Num. of linear elements=1503 Num. of nodes=1570	
View mesh Close Figure, 16 Mesh information, GiD will inform the user about the quality (of the mesh created.

Running test

Arteries 1D was thought and realized in order to make simple the test performing. In the Bar menu there is a sub-menu named *calculate*. From this the user has to choose the voice *calculate* in order to perform the test. If the user set up correctly the model, Arteries 1D will start the computation. Using the tool View process info...

It is also possible to visualize some information related to the computational process, like the time needed to calculate the case or the state of the computation. If some error occurs or the solver do not converge warning or error messages will be visualized in the same window



Results analysis

When all the calculations are finished the user may want to analyze the test just run.

User will charge the file containing the results through the tool bar. The kind of results that will be possible to open was be decided in the preprocess (see fig. 6).

Gill GiD x64 Project: UNNAMED	
Fries View Other New Ctrl-n Image: Ctrl-n Image: Ctrl-n Image: Ctrl-n Image: Ctrl-n	Firstly the user has to pass
Openmalipie Ctri-o	to the post processor
Merge Recent post files	program. GiD
Import Export	willautomatically charges
Preprocess Becent projects	every setting only using the
Print to file	icon showed in the blue
Page and capture settings	circle.
Quit Ctrl-q	
	After that the user has to
60 Postprocess Read	open the postprocess file
Directory: SPA2.gid	results with Ctrl+O or using
Eul_CPA2_gen.flavia.dieplacement.res E Full_SPA2_gen.flavia.Flow.res	the menu like showed in
Er ull_SPA2_gen flaviourish No Preview E Full_SPA2_gen flavia pressure res E Full_SPA2_gen flavia velocity res	red circle.
File name: Full SPA2 gen.flavia.Flow.res	
Files of type: GiD postprocess (*.res,*.msh,*.bin)	A classic window will
	appear and the user will
	choose the file . <i>res</i> that will
	want to see and analyse
Figure, 18 How to charge a post process results	; file

The charged results may be analyzed in several modes:

- 1. Flow vs Time points evolution: GiD plots a graph of one or more points showing how the flow will change all along the simulation time
- 2. **Pressure vs Time points evolution**: GiD plots a graph of one or more points showing how the pressure will change all along the simulation time

- 3. Velocity vs Time points evolution: GiD plots a graph of one or more points showing how the velocity will change all along the simulation time
- 4. **Displacement vs Time points evolution**: GiD plots a graph of one or more points showing how the displacement will change all along the simulation time

A lot other tools are available in GiD, in order to perform a more complete and exhaustive analysis of the run tests. For more information user can consult the web page <u>www.gidhome.com</u> and download example and tutorial that will explain the functionalities and the tools of GiD in the processor or post processor.