ADFC

Navier-Stokes solver

English tutorial

June 2003

http://adfc.sourceforge.net

Introduction

This brief tutorial will introduce you to using **ADFC** solver and **GiD** pre/postprocessor. It is not intended to be an exhaustive document about GiD: for further information check the CIMNE web page[10], where you will find many information.

As we are continuously improving software and documentation, please consider this as an "evergreen" document and check regularly our web page for new versions of this tutorial.

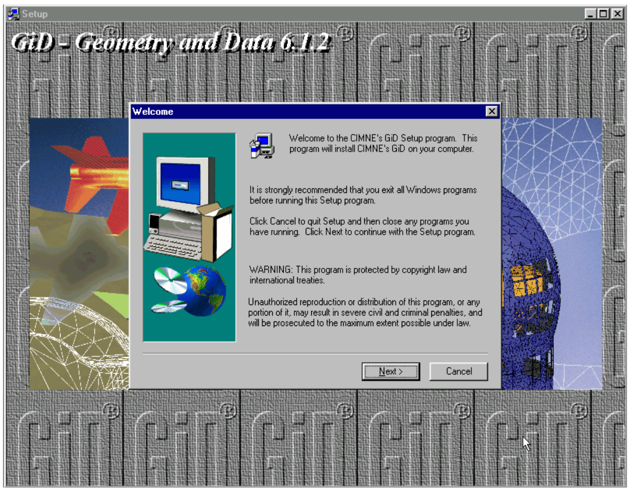
Don't hesitate to sending doubts or request to:

ADFC Team list	adfc-devel@lists.sourceforge.net

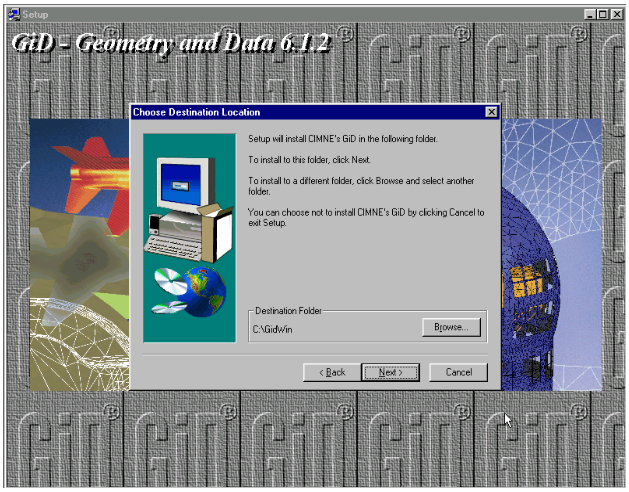
10 http://www.cimne.upc.es

GiD installation

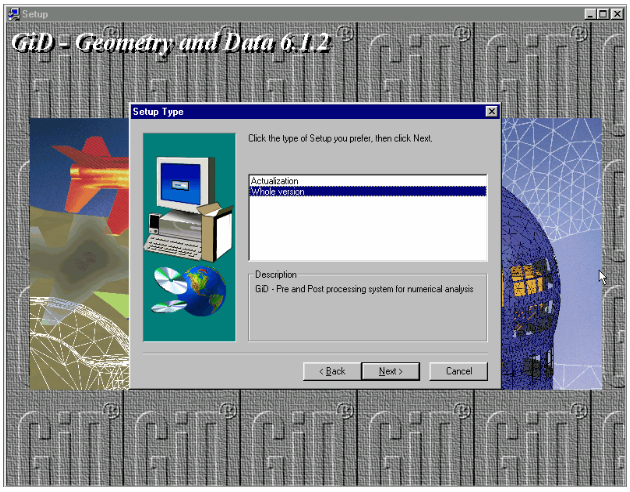
In order to install GiD in your computer, you must download the installation file from CIMNE webpage. GiD is commercial software, so you will need a license to use it but, in spite of that, you can obtain from its web page a 30-days free evaluation license. ADFC team is not related with CIMNE in anyway: we are willing to make our solver compatible with as many meshers as possible, so if you know a mesher that would fit for your purposes, let us know and provide us with enough documentation to work on compatibility.



1. *GiD is commercial software but it can be installed and used for a 30 days evaluation period. In order to do so, you will need a license that can be freely downloaded from its web page.*

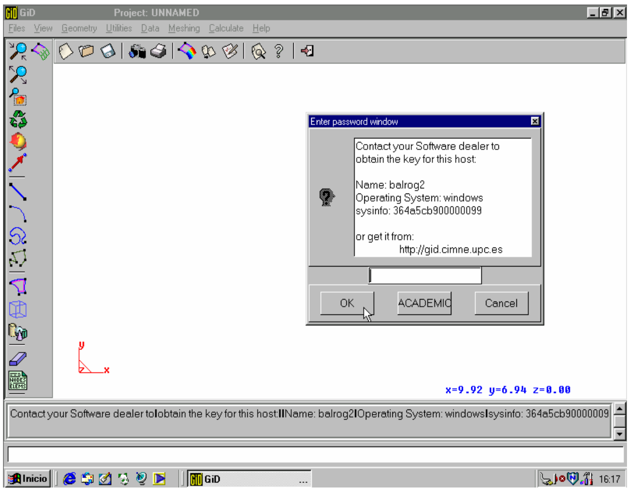


2. *After license agreement, you will be prompted for an installation folder. Remember this directory! You will need it to install ADFC.*

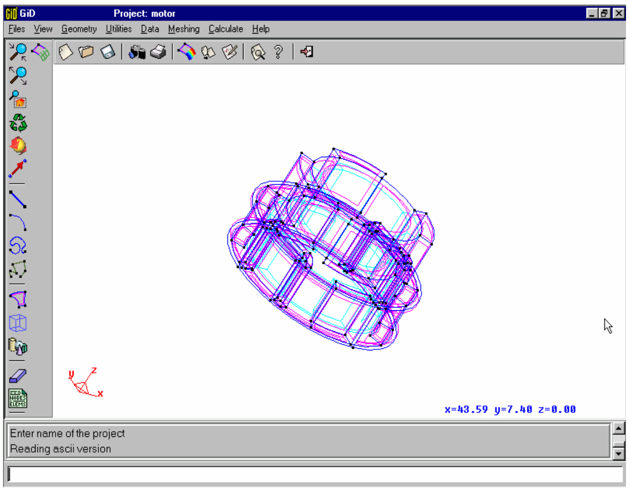


3. *You have to choose complete installation*

4. *Afterwards, file copy will begin.*



__5.__ When it is executed for the first time, you will have to introduce the serial number you got from CIMNE webpage.



__6.__ GiD is ready for use! You can see an example we have just loaded.

ADFC v3.x Installation

In order to install ADFC you need to download from Sourceforge[11] the most recent zip file available. This file contains a folder named *adfc.gid* that you have to extract into *ProblemTypes* GiD subfolder.

This folder contains a bundle of auxiliar files that are needed to communicate GiD with the solver. As soon as you extract the content of the file to the proper path, GiD will recognize it and add an specific entry to its menu "**Data/Problem Type/adfc**".

On the other hand, the *bin* folder contains the ADFC executables. You should choose the right one for your platform and operating system.

Compiling ADFC v3.x

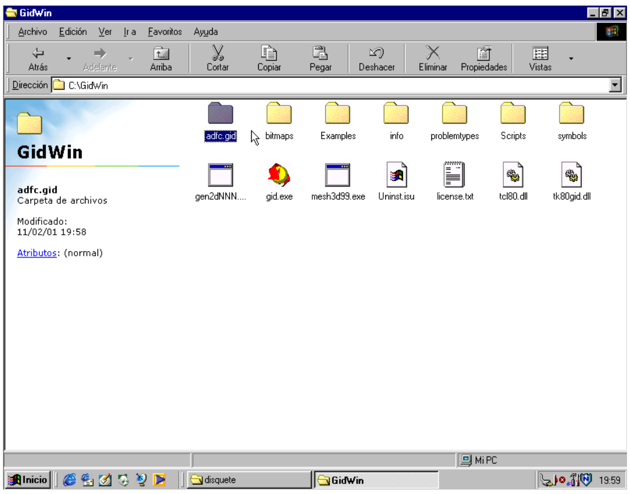
If you wish to use ADFC on a platform not supported by us or you have applied changes to its source code, you will need to recompile it. As ADFC is written in standard C++ and it is not linked against any uncommon library, everything should go smoothly.

In order to compile the program under UNIX or Linux, open a shell, browse to the *src* folder and run the command *make*. This will generate an executable for your platform. In very strange cases you will need to modify the *makefile* manually. Please, don't hesitate to contacting us about any problem or doubt you have: we are very interested in making ADFC as robust and portable as possible.

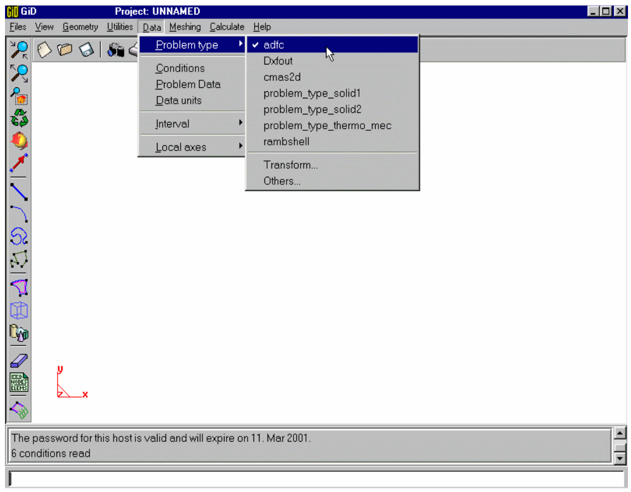
11 La ficha técnica se encuentra en http://www.sourceforge.net/projects/adfc

Solving an easy case: 2D cylinder at Reynolds 100

At this point, the solver and the mesher are properly installed on your sistem. Below it is described step by step the **preprocess**, **solving** and **postprocess** of an easy case: the well-known 2D cylinder at Reynolds 100.



*Firstly, let's check that the **problem type** is correctly installled inside GiD directory.*



*The first step is to indicate GiD that you wish to use the ADFC problem type, in the menu "**Data/Problem Type/adfc**"*

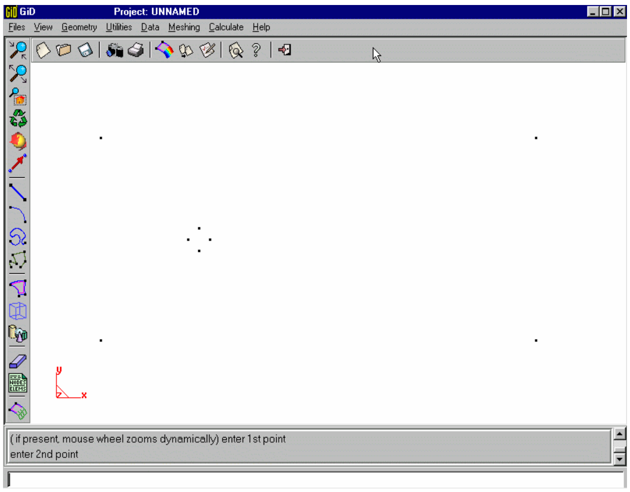
The first step is to indicate GiD that you wish to mesh a geometry for ADFC solver, choosing in the menu “**Data/Problem Type/adfc**”. Afterwards, you are going to define the geometry through the mouse and the keyboard, introducing point coordinates, lines and surfaces.

The cylinder has a 1 unit diammeter and will be placed in the coordinate origin. The domain will be a rectangle 20 units long and 9 units wide. Therefore, you will need to define all these points:

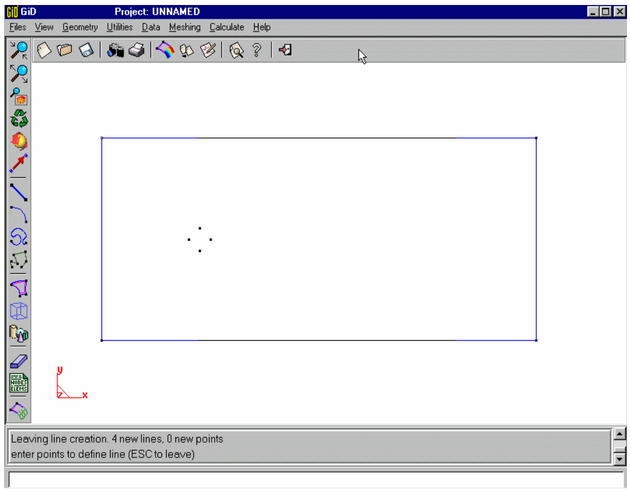
X coordinate	Y coordinate
-0.5	0
0	0.5
0.5	0
0	-0.5
-4.5	4.5
15.5	4.5
15.5	-4.5
-4.5	-4.5

Point creation is performed by selecting in the menu the option **"Geometry/Create/Point"**. Points can be introduced through the keyboard, just typing their X and Y coordinates delimited by whitespaces and pressing *enter* key after each one. This can also be performed using the mouse and clicking directly on the window, but it is extremely unaccurate and we disencourage that.

The next step is to connect the points with straight lines and define the rectangular boundary. You can accomplish it with the menu option **"Geometry/Create/Line"** and by clicking on the 4 vertex of the rectangle. When you are finished, press *scape*.



As you introduce the coordinate values, points will appear in the screen.



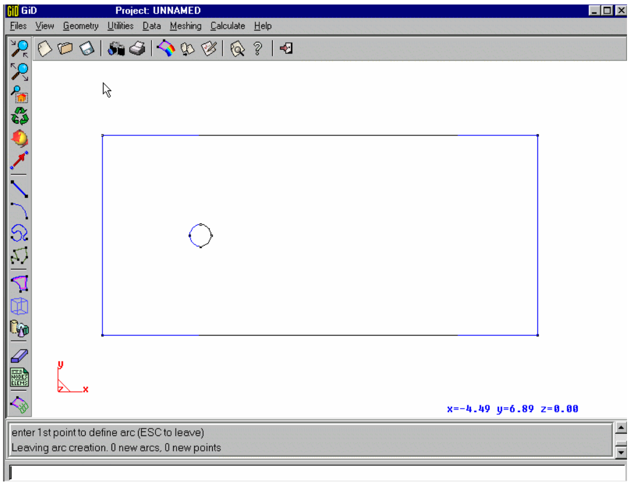
*With the option "**Geometry/Create/Line**" you can define the straight lines that surround the domain.*

Cylinder creation is a bit more complicated. Even though circular shapes can be easily created with a predefined function in GiD, this tutorial will show how to use curvilinear arcs so you will learn how to face more complex geometries which may not be predefined in the mesher. Choose the option **"Geometry/Create/Arc"**which allows you the define an arc with three points. Cylinder will consist on two semicircunferences: one right-up-left and the other left-down-right. GiD will always show a brief help text in the status bar.

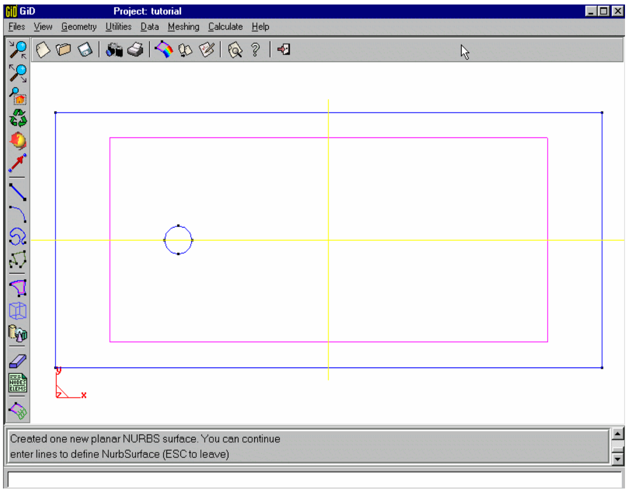
At this point, you have the boundaries properly defined, so the next step is to tell GiD how the domain is. As there is a hole in it (the cylinder) you will define the rectangle firstly and then sustract the cylinder from it.

Rectangle is defined through the option **"Geometry/Create/NURBS Surface/By Contour"** highlighting the four lines that define the region and pressing *scape*. A new pink rectangle will appear indicating that the region was successfully created.

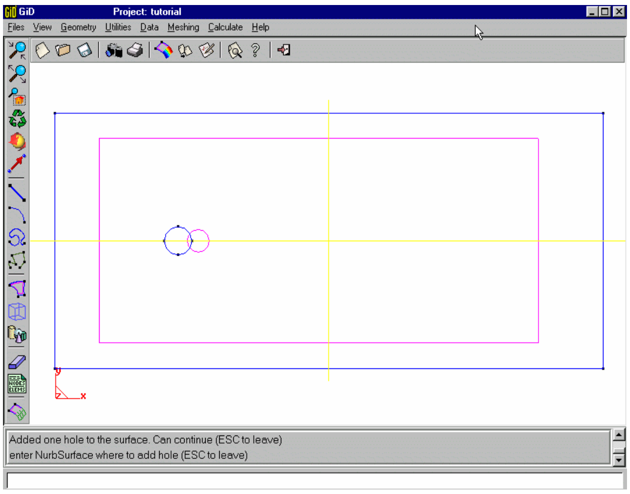
In order to add the hole, choose **"Geometry/Edit/Hole NURBS Surface"** and select the previously defined rectangular region (the pink rectangle) and then the two arcs that define the hole. Pressing *scape* will update the region and the hole will be added to it.



You have just defined al geometry lines.



Firstly, define the rectangular region.



After substracting the cylinder from it, the geometry is completely defined.

At this point, geometry is perfectly defined so preprocess is finished. Now you will define boundary conditions, meshing parameters and some solver specific configuration.

GiD can mesh using several types of elements. ADFC uses Taylor-Hood triangles in 2D cases, so you will need to choose int the menu **"Meshing/Quadratic Elements/Quadratic"** and **"Meshing/Element Type/Triangular"**.

Just before generating the mesh, you have to set **boundary conditions**: where the fluid enters the domain, which is the downstream exit, which walls are rigid and so on. For this case, the fluid enters from the left side, with an homogeneous speed profile and exits through the opposite side on the right. The upper and lower walls, as well as the cylinder, will be static rigid walls, so you will set on them zero speed.

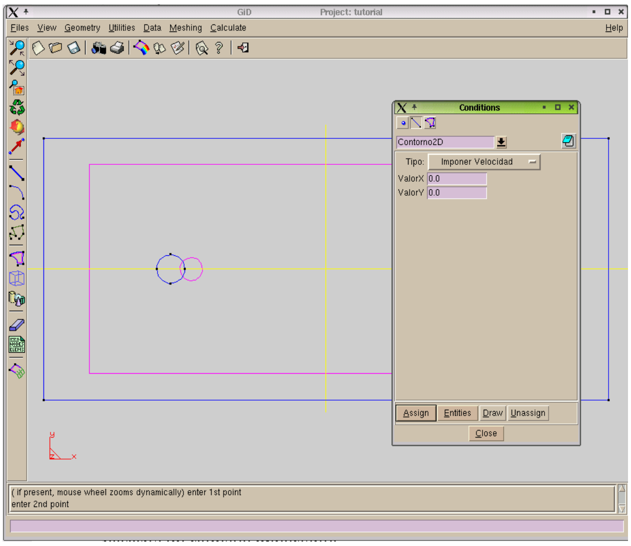
Open the **Conditions** dialog window by selecting it from the menu **“Data/Conditions"**. On the top of the dialog there are three icons, representing boundary conditions over nodes, lines and surfaces. You need **line** conditions in this 2D case.

Type	**X Value**	**Description**
Set_speed	1.0	*Fluid entrance, homogeneous speed profile (1,0)*
Set_speed	0.0	*Upper wall*
Set_speed	0.0	*Lower wall*
Set_speed	0.0	*Cylinder*
Downstream NBC	0.0	*Fluid downstream exit*

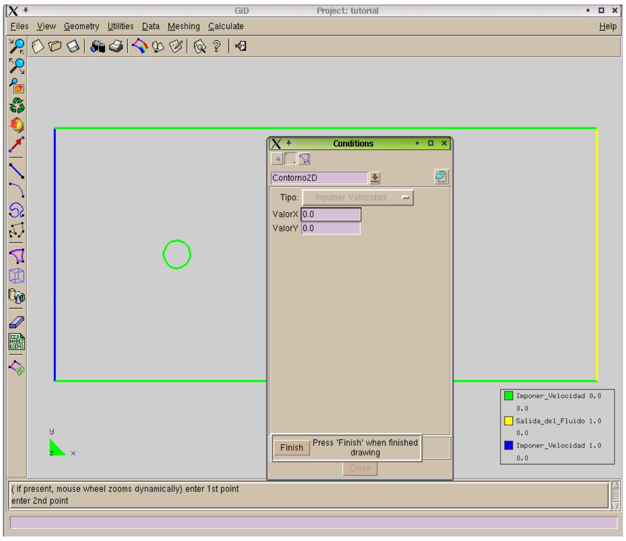
Set_speed type applies a Dirichlet condition to the speed field. Speed on that places will be equal to the values you indicate in the dialog. **Downstream_NBC** type is set on the exit of the fluid so it can pass through that place without tension.

In order to assign a condition to a line, you have to click the button **"Assign"**. After pressing it, you can choose all the lines you wish to add to this boundary type. When you are finish just press *scape*.

You can check that all conditions were set properly by choosing the option **"Draw/Draw Conditions Field/Type”** in the dialog.



*The menu item "**Data/Conditions**" open the dialog window used to set boundary conditions.*

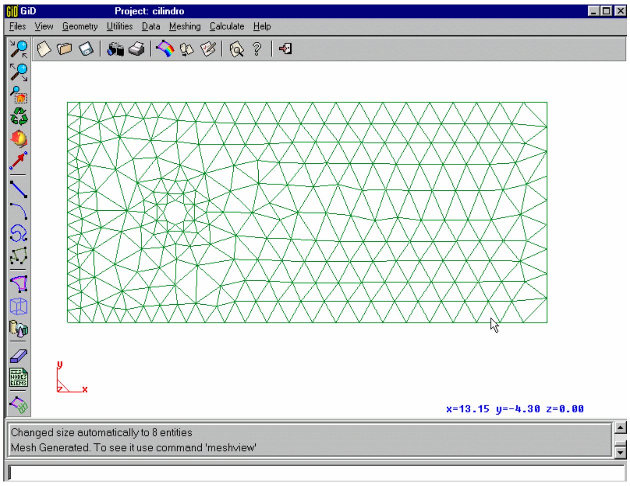


*You can check if the conditions were properly set by choosing "**Draw/Colors**" .*

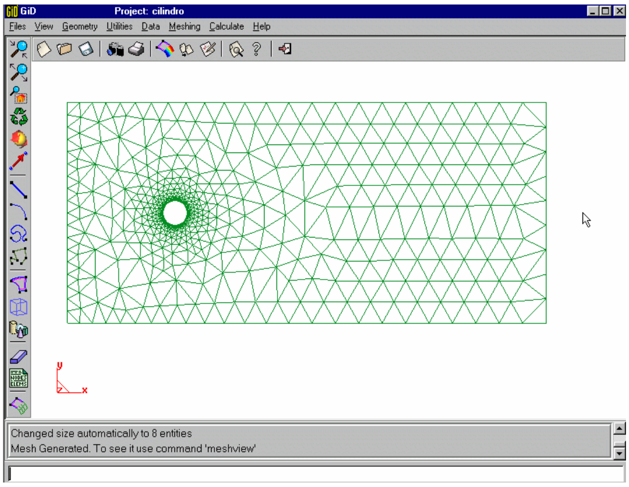
Only mesh generation is left. If you choose **"Meshing/Generate"** in the menu you will generate a new one. GiD will ask you to confirm that you wish to overwrite the previous one (yes, of course) and will prompt you for the average element size. Let's use the default one: as you can see, the generated mesh is too coarse and requires tunning.

In order to tell GiD which places require more detailed meshing select **"Meshing/Asign unstruct. sizes/Lines"** and introduce a 0.05 size in the dialog. When GiD asks for which lines to assign that mesh size to, select the boundaries of the cylinder and press *scape*. If you mesh again, with **"Meshing/Generate"** and the same default element size, mesh will have a better look. The zoom (magnifying glass icon) you can see the mesh in greater detail.

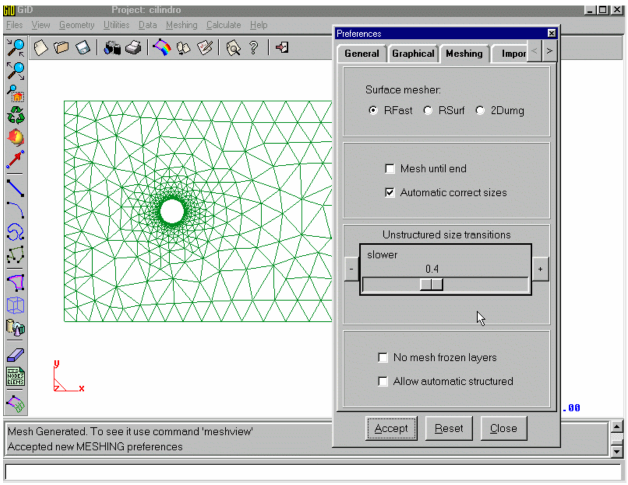
Now let's force GiD to perform smoother element size transitions in the mesh. Select **"Utilities/Preferences"** which opens a dialog window and then click on the **"Meshing"** tab. There you will find an horizontal scrollbar labeled **"Unstructured size transitions"**. This value tells GiD how fast element size can change. Choose a smoother one, a value like 0.4 will be a good choice. Meshing again you will get the nice result you were looking for.



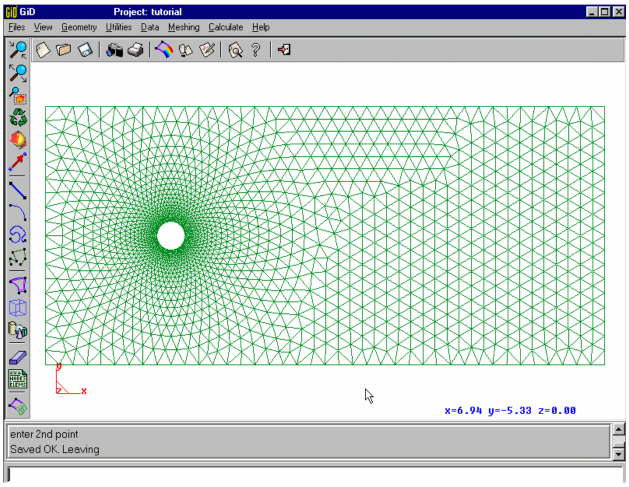
Default meshing parameters may result on a very coarse mesh.



Assigning to the cylinder a smaller element size will improve the mesh around it.



Afterwards we can force GiD to perform smoother element size transitions.

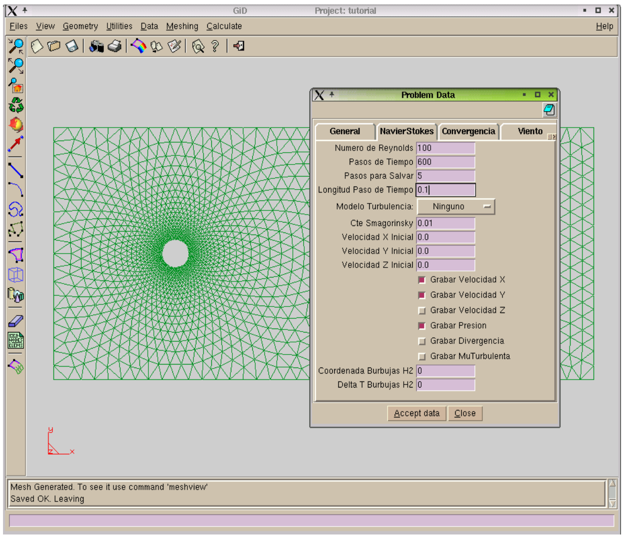


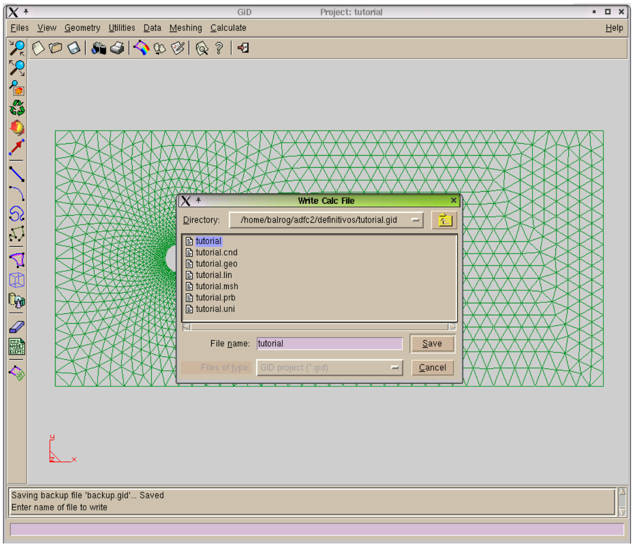
The final mesh! 7502 nodes and 3662 elements. Don't worry if you don't achieve this values... it should be fine if your mesh looks more or less like this one.

Preprocess and meshing is finished! Just before launching the solver, you will need to configure some CFD specific values in the menu **"Data/Problem Data"**. Those values are passed directly to ADFC by the GiD. Default values will fit well, but check these important ones:

Tab	Parameter	Value
NavierStokes	Reynolds_number	100
NavierStokes	Time_steps	2000
NavierStokes	Steps_between_saving	10
NavierStokes	Time_step_length	0.05

Let's save your work with the menu item **"Files/Save"** (GiD format). If possible, don't use a filename longer than 8 characters (this limitation only affect some older version of GiD running under Windows). *"tutorial"* would be fine, and a folder named “*tutorial.gid*” will be created, containing several archives in it. As ADFC cannot read this file format, you will need to generate a *Calculation File*. In order to do so, click in the menu “**Files/Import Export/Write Calculation File**” and create a file called “*tutorial*” with no extension in the previously used folder “tutorial.gid”.





Solver parameter configuration.

Saving Calculation File.

Solver is launched within a terminal or console. In the command line, just write the following command:

<**adfc3.exe | adfc3**> **-gid** <*path_to_calculation_file*>

For example, Windows platform:

c:\gid61\adfc.gid\adfc3.exe -gid c:\mallados\tutorial.gid\tutorial

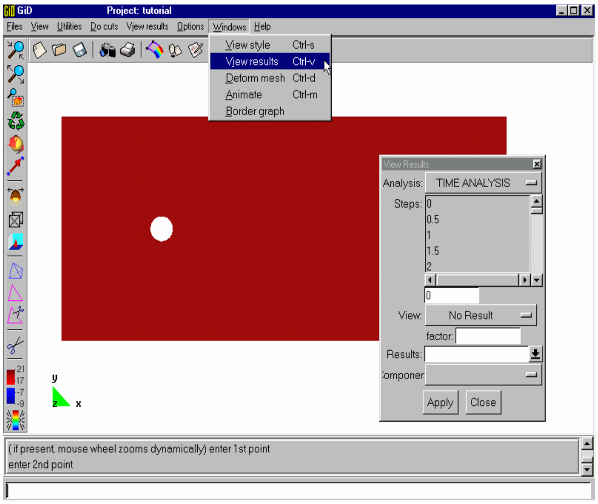
Under Linux:

/home/user/adfc3.exe -gid /home/user/tutorial.gid/tutorial

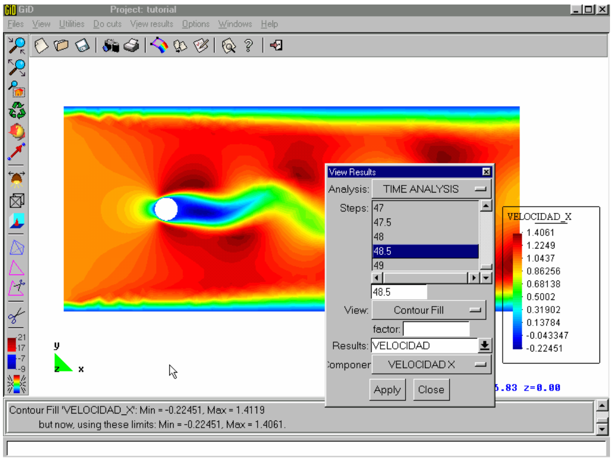
Depending on your hardware, solving may take more or less time to complete. You can abort a solving process when you like, pressing *Ctrl-C* in the console, but you will be able to posprocess a running one if you like.

Postprocess is started by selecting **"Files/Postprocess"**. A dialog window will appear while data is loaded from disk.

In **"Windows/View Results"** you can choose which instant or step to visualize, which *view* to use (choose *Contour Fill*), which result (*SPEED*) and component (*SPEED_X*, for example).



"View Results" allows you to choose which time step to visualize, view, result, component...



Speed X component, at time 48.5 seconds.

GiD and ADFC have huge capabilities... this is just a small demo of them. We encourage you to check ADFC documentation and GiD webpage.