The Focus

A Publication for ANSYS Users

Contents

Feature Articles

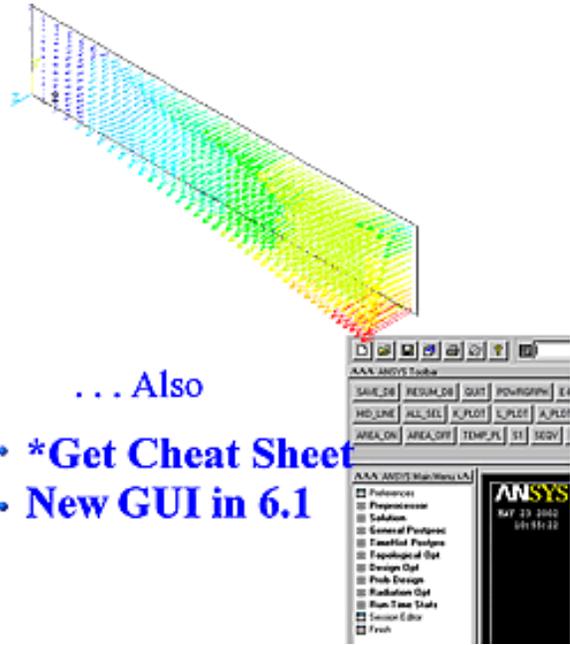## On the Web

Resources

. . . Also

- *Get Cheat Sheet
- New GUI in 6.1

ANSYS Support Distributor

A Publication for ANSYS Users

FLOTRAN   Non-Uniform Velocity Distribution

By [J. Luis Rosales](), Thermal and Fluid Analyst

The purpose of this article is to demonstrate the use of macros for applying non-uniform velocity distributions on the boundaries of a domain for a computational fluid dynamics (CFD) simulation using FLOTRAN. Although the user can apply a varying velocity on a boundary using a table or the function generator in ANSYS, this article will illustrate the simplicity and flexibility provided by using a macro. For many CFD problems, it may be fortunate that the application of a uniform velocity profile at the inlet of a domain does not affect the results of interest. However, there are situations where the user absolutely must apply a velocity distribution that is not uniform. With this in mind, two example problems are used to teach the user how to vary the velocity on a boundary. The first example problem that will be shown is the flow past a backward-facing step. The problem will show the ease of applying a non-uniform and a parabolic inlet velocity profile. The second example problem will show users how to apply a radially varying velocity on the lower boundary of an axisymmetric, rotating disk in a cavity.

Example Problem 1: Backward-facing Step Flow

A schematic of the problem domain is shown below in Figure 1. The domain is divided into two regions to simplify the application of a varying velocity boundary condition on the upper, left-hand inlet boundary. The long arrows on the upper and lower walls and on the lower, left-hand wall depict the velocity vectors that are set to zero to simulate a no-slip boundary condition. The outlet boundary condition on the right-hand side of the domain is specified with a constant pressure of zero. The last task is applying the non-uniform velocity distribution on the upper, right-hand side boundary (depicted by the small arrows).
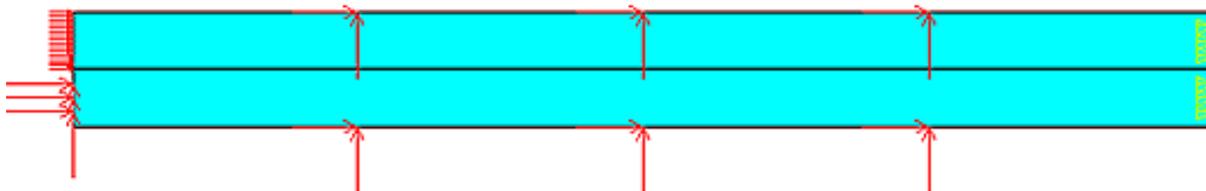

Figure 1. Schematic of the backward-facing step domain and applied boundary conditions.

The application of a linear and a parabolic velocity distribution on the required line is accomplished with the use of the macros shown in Figures 2 and 3, respectively. In order to use either macro, the user must first isolate the line on which the desired velocity profile is to be applied. This can be accomplished with the line select logic using the entity select menu. The macros will then select the nodes associated with the selected line and apply the velocity distribution.

The Focus

A Publication for ANSYS Users

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
! APPLY A LINEAR VELOCITY DISTRIBUTION TO NODES ATTACHED TO !
! SELECTED LINES. THE DISTRIBUTION IS IN THE Y-DIRECTION.   !
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!
! ENTER THE MINIMUM AND MAXIMUM VELOCITIES
!
 VMIN = 0.0
 VMAX = 0.5
!
 NSLL,S,1
*GET,CNT_SEL,NODE,,COUNT,,,
*GET,NODE_NUM,NODE,,NUM,MIN,,
!
*GET,Y_MIN,NODE,,MNLOC,Y,,
*GET,Y_MAX,NODE,,MXLOC,Y,,
!
 SLOPE = (VMAX - VMIN)/(Y_MAX - Y_MIN)
*DO,I,1,CNT_SEL
   VV = VMIN + (NY(NODE_NUM) - Y_MIN)*SLOPE
   D,NODE_NUM,VX,VV
   D,NODE_NUM,VY,0.0
   *GET,NODE_NUM,NODE,NODE_NUM,NXTH,,,
*ENDDO
```
Figure 2. Macro for the application of a linear velocity distribution on a line.

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
! APPLY A PARABOLIC VELOCITY DISTRIBUTION TO NODES ATTACHED !
! TO SELECTED LINES. THE DISTRIBUTION IS IN THE Y-DIRECTION.!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!
! ENTER THE AVERAGE VELOCITY
!
 VAVG = 0.5
!
 NSLL,S,1
*GET,CNT_SEL,NODE,,COUNT,,,
*GET,NODE_NUM,NODE,,NUM,MIN,,
!
*GET,Y_MIN,NODE,,MNLOC,Y,,
*GET,Y_MAX,NODE,,MXLOC,Y,,
!
 HGT = Y_MAX - Y_MIN
!
*DO,I,1,CNT_SEL
   TEMP1 = (NY(NODE_NUM) - Y_MIN)
   VV = (6.0*VAVG/HGT**2)*(HGT*TEMP1 - TEMP1**2)
   D,NODE_NUM,VX,VV
   D,NODE_NUM,VY,0.0
   *GET,NODE_NUM,NODE,NODE_NUM,NXTH,,,
*ENDDO
```
Figure 3. Macro for the application of a parabolic velocity distribution on a line.

The macro in Figure 2 requires the user to specify the minimum and maximum velocities for the variables VMIN and VMAX, respectively. Note that these values can easily be exchanged to reverse the linear velocity profile on the boundary. Figure 4 shows the linear velocity distribution created by the macro in Figure 2.
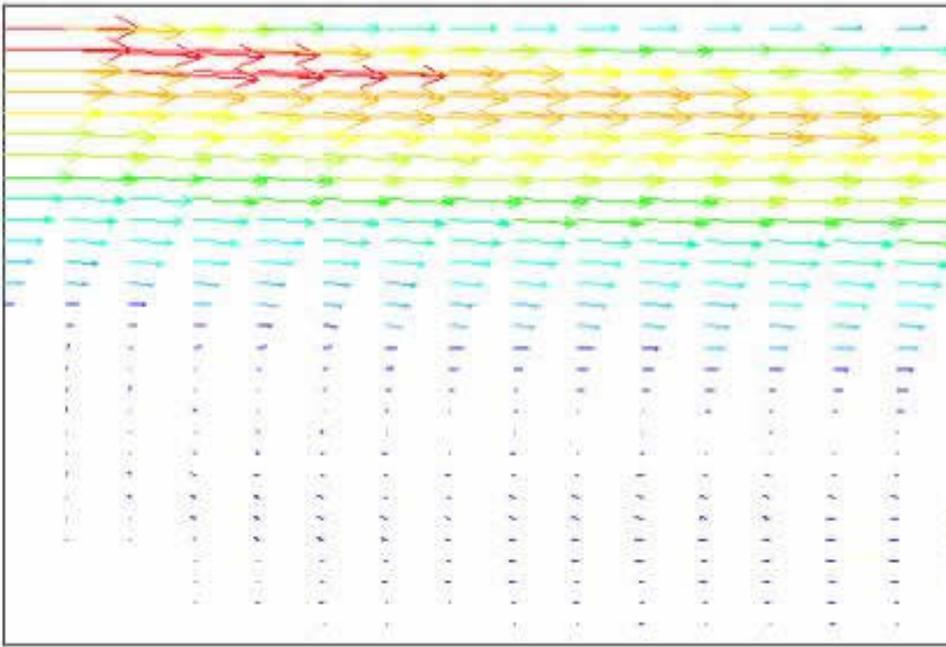
The Focus

A Publication for ANSYS Users

Figure 4. Linear velocity distribution at the inlet of the backward-facing step.

Figure 5 shows the parabolic velocity profile created by the macro in Figure 3. Note that the user only needs to enter the average velocity using the variable VAVG. The DO loop contains the equation to create a parabolic distribution. The macro serves as a template for a user to type in any equation to produce any desired velocity distribution.
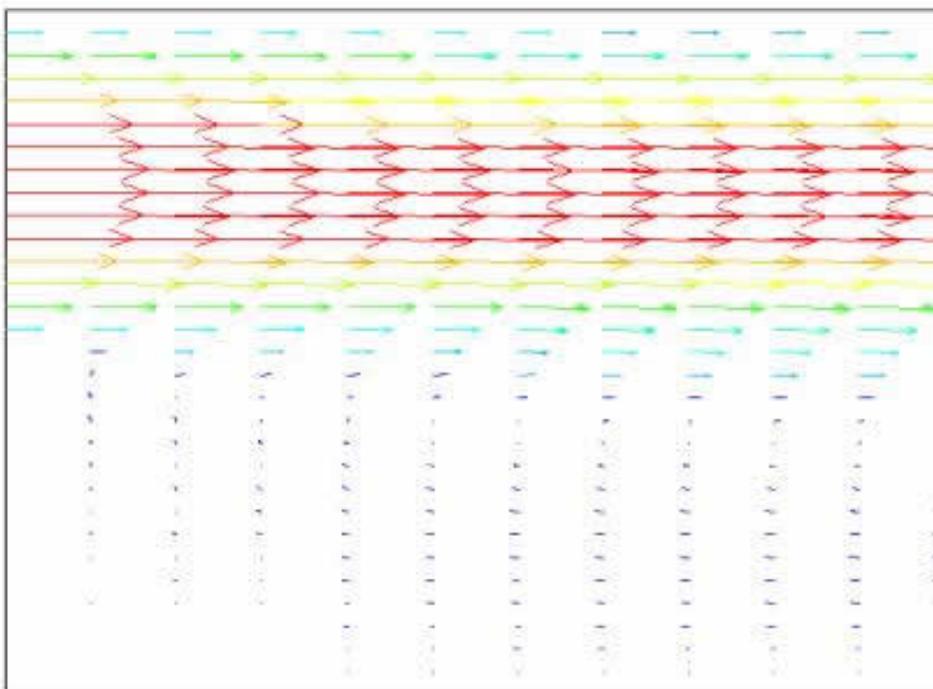

Figure 5. Parabolic velocity distribution at the inlet of the backward-facing step.

A Publication for ANSYS Users

Figures 6 and 7 show the complete velocity fields produced by linear and parabolic velocity profiles, respectively. This variation in the velocity field can produce temperatures of components on the walls that vary significantly and thus produce inaccurate results.
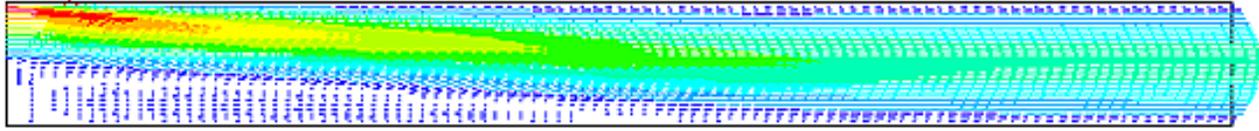

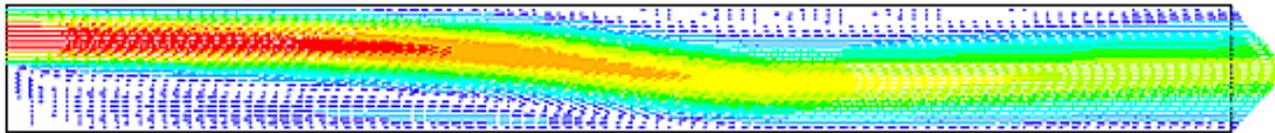Figure 6. Complete velocity field for a linear velocity distribution.

Figure 7. Complete velocity field for a parabolic velocity distribution.

Example Problem 2: Confined, Axisymmetric Disk Flow

A schematic of the problem domain is shown below in Figure 8. The two side boundaries and upper boundary (depicted by the long arrows) are stationary walls with an imposed zero velocity. The lower line with the small arrows is the spinning disk boundary with the axis of symmetry along the left vertical line.
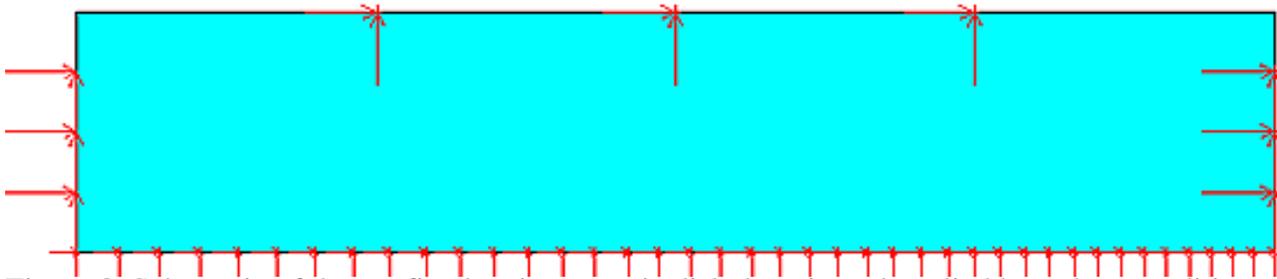

Figure 8. Schematic of the confined, axisymmetric disk domain and applied boundary conditions.

The application of a radial velocity distribution on the lower boundary of the disk domain is accomplished with the use of the macro shown in Figure 9. The line on which the varying velocity will be applied needs to be isolated using the select login provided by ANSYS. Once this is done, the macro can be read into ANSYS and the desired velocity distribution will be applied on the selected line. Note that the macro, as shown in Figure 9, is valid for axisymmetric models about the y-axis only. The macro can easily be modified for models that are axisymmetric about the x-axis.

The Focus

PADT

A Publication for ANSYS Users

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
! APPLY A TANGENTIAL VELOCITY DISTRIBUTION TO THE NODES ATTACHED   !
! TO THE SELECTED LINES.  MODEL IS AXI-SYMMETRIC ABOUT THE Y-AXIS.!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!
 RPM = 10000
 ALLSEL,BELOW,LINE
 NSLL,S,1
*GET,CNT_SEL,NODE,,COUNT,,,
*GET,NODE_NUM,NODE,,NUM,MIN,,
*DO,I,1,CNT_SEL
   D,NODE_NUM,VX,0
   D,NODE_NUM,VY,0
   D,NODE_NUM,VZ,RPM/60*2*ACOS(-1)*NX(NODE_NUM)
   *GET,NODE_NUM,NODE,NODE_NUM,NXTH,,,
*ENDDO
```

Figure 9. Macro for the application of a radial velocity distribution on a line.

Instead of having the user specify the maximum and minimum velocities along the length of the line, the macro has been set up so that the RPM is entered. The RPM is set to 10,000 and can be changed to any angular rotation desired. Figure 10 shows the velocity distribution obtained for the confined, axisymmetric disk problem.

Figure 10. Three-dimensional velocity distribution for confined, axisymmetric disk flow configuration.
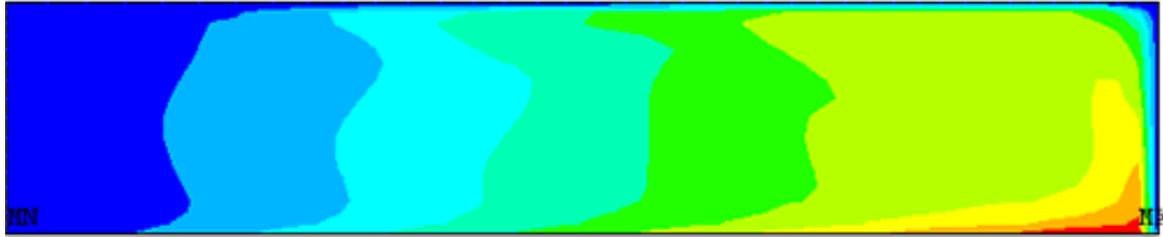
A Publication for ANSYS Users

Figure 11. Secondary flow distribution for the confined, axisymmetric disk flow.

Note, it is very important that the user set the axisymmetric option for the Fluid141 elements. The rotation of the disk causes a secondary flow field to develop, Figure 11, because of the centrifugal forces produced by the rotation.

Conclusion

The two examples introduced in this article demonstrate the ease with which a user can apply a complicated velocity boundary distribution for any problem. The macros also serve as a template for users to build more complicated velocity distributions. With only minor modifications, the macros can be extended to three-dimensional situations.

*GET Cheat Sheet

by [Rod Scholl](#)

Using the help files for the `*GET` command can be laborious. We have compressed the data into a two-sided 8.5×14" cheat sheet. This sheet also includes the `*V` functions, `*VGET`, the `GET` functions (such as `NX()`, `TEMP()`, etc.) and a few other miscellaneous items.

You can print out your own copy from the following Adobe Acrobat file:

[get_sheet.pdf](#) (364 KB)

If you have trouble printing, or want it on more durable paperboard, [send me](#) a note, specifying how many you would like, and we will mail you hardcopies *while supplies last* no guarantees!

A sample image is shown below to give you a feel for what it looks like.
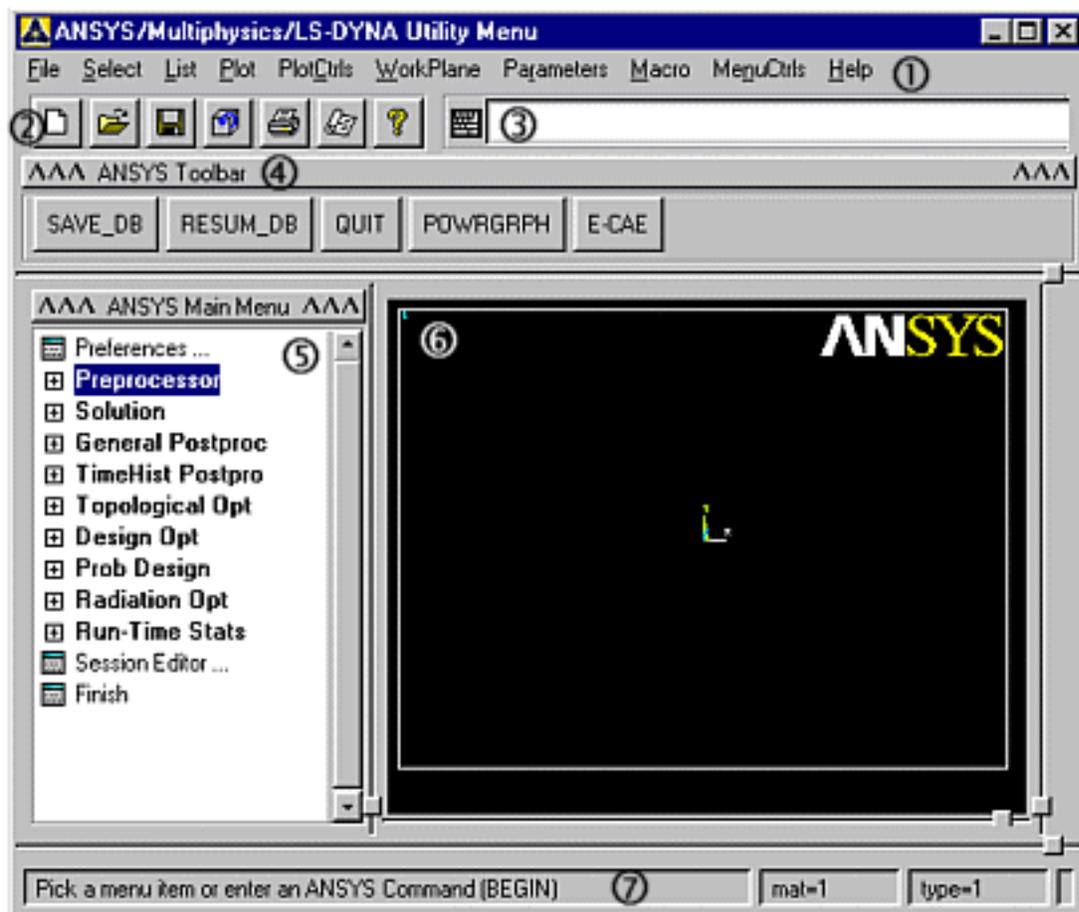

Sample of the *GET Cheat Sheet.

A Publication for ANSYS Users

ANSYS 6.1 New GUI

by Rod Scholl

If you ve already installed ANSYS 6.1   then this will be old news. If you haven t,
then take a look at the new GUI introduced in ANSYS 6.1.

Everything except the output window is now in one interface. There is a tree
hierarchy used for the Main menu.

There are several reasons for the change. Primarily, updating the GUI using ~tcl
adds a new level of customization possibilities for ANSYS. Previously, one
would need to learn UIDL, the ANSYS-designed menu scripts to make minor
changes to the layout. Now one can use platform independent ~tcl to fully
customize the program interface for one s own company application.

There are a few things that work differently than the old GUI   windows not

popping up where we expect, or not minimizing the way it used to. Although these differences can be irritating, they can be altered in future releases. Everyone likes application windows to work slightly differently. So, we all might as well learn it now and steer it the way we like it. [Send me](mailto:) or your ASD your recommendations and beefs!

And Finally:

If you want a reprieve, you can activate the old menus   but no telling how long these will be available. It is said that they may be gone as soon as 6.2!

To return to the pre-ANSYS 6.1 GUI, enter `/MSTART, UTIL,1` in your `start61.ans` file.

A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don t hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to   or unsubscribe from   *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.