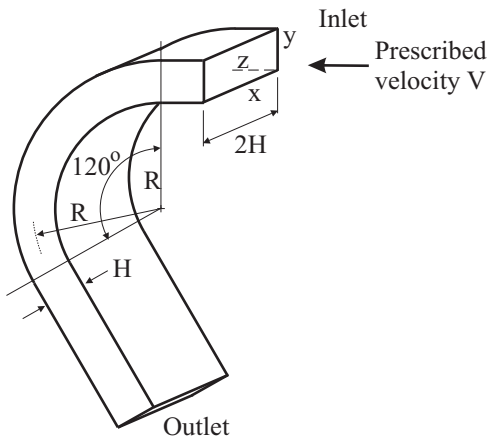
Problem description

It is desired to analyze the flow in the strongly curved channel shown.



Numerical data for this model is $H = 0.127$, $R = 3H$, $V = 3$. Laminar fluid material constants are $\rho = 1000$, $\mu = 1.27 \times 10^{-3}$, giving a Reynolds number of $\mathrm{Re} = \dfrac{\rho \mathrm{VH}}{\mu} = 3 \times 10^5$.

This problem is also considered in the following references:

Brunn, H.H., "An Experimental Investigation of Secondary Flow Losses in Bends with Rectangular Cross Sections," CUED/A-Turbo/TR 95, Cambridge University.

Khalil, I.M and Weber, H.G., "Modeling of Three-Dimensional Flow in Turning Channels", *Transactions of the ASME - Journal of Engineering for Power*, paper 84-GT-49, 1984

In our numerical model, we use the k-ε turbulence model. Turbulence variables k and ε are specified at the inlet according to the following formulas:

$$k_{inlet} = \frac{3}{2}(i \cdot V_{inlet})^2, \quad \varepsilon_{inlet} = \frac{\left(k_{inlet}\right)^{3/2}}{0.3D}$$

where i is the turbulence intensity, here set to 0.025 and D is the hydraulic diameter, here equal to H.

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Turbulence material modeling
- Turbulence boundary conditions and initial conditions
- Using the FCBI-C elements
- Restart to second order FCBI-C elements
- Mass flux calculation using element face variables

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem cannot be solved with the 900 nodes version of the ADINA System because the model contains 13041 nodes.

Much of the input for this problem is stored in files `prob42_1.in`, `prob42_2.in`, `prob42_1.plo`, `prob42_2.plo`. You need to copy files `prob42_1.in`, `prob42_2.in`, `prob42_1.plo`, `prob42_1.plo` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA CFD.

Defining model control data

Flow assumptions: Choose Model→Flow Assumptions, uncheck the Includes Heat Transfer button, set the Flow Model to Turbulent K-Epsilon and click OK.

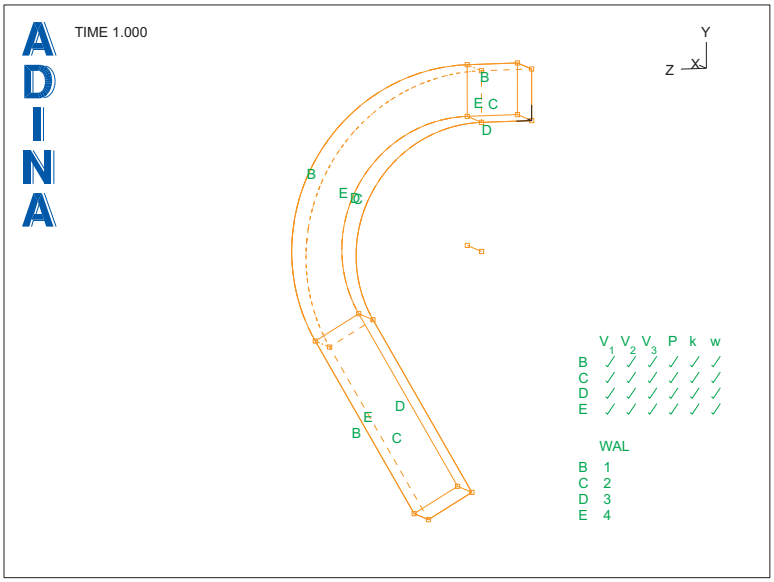
Element type: Choose Control→Solution Process and set the 'Flow-Condition-Based Interpolation Elements' to FCBI-C. (Do not close the dialog box yet.)

Outer iteration settings: Click the Outer Iteration… button. Now click the Advanced Settings… button, set the Equation Residual to "Use All", the Equation Residual Tolerance to 1E-5, the Variable Residual Tolerance to 1E-4, the "Interpolation Scheme for Pressure" to Linear, check the "Use Pressure-Implicit with Splitting of Operators (PISO) Scheme" button, then click OK three times to close all dialog boxes.

Defining the geometry and special boundary conditions

We have prepared a batch file (`prob42_1.in`) which contains the model geometry and special boundary conditions.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob42_1.in` and click Open. The graphics window should look something like this:



Only half of the flow domain is modeled. Special boundary condition 4 (marked with an E on the plot) is a slip-wall boundary condition. This boundary condition is chosen to model symmetry.

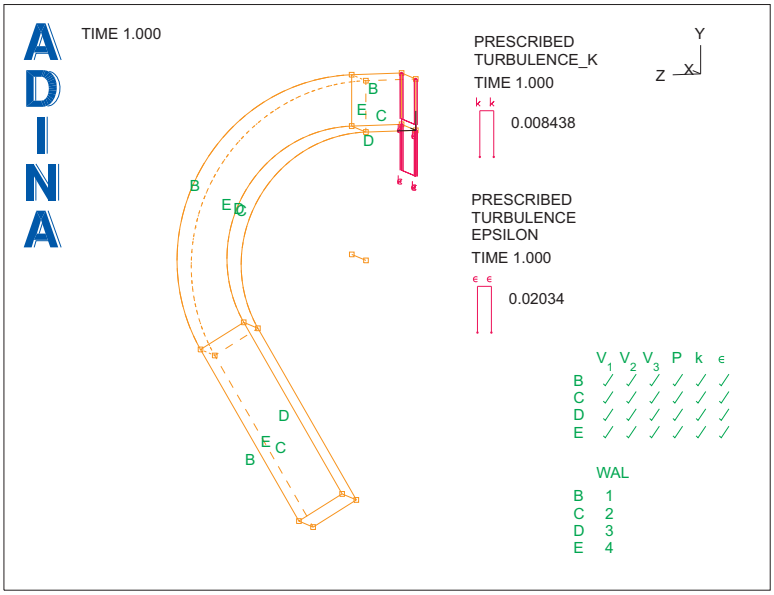
Defining the turbulence data

Turbulence material: Click the Manage Materials icon and click the K-Epsilon Standard/RNG Model button. In the Define Turbulent K-Epsilon Material dialog box, add material 1, set the Laminar Viscosity to 1.27e-3 and set the Density to 1000. We will not change the defaults for the Turbulent Model Flow Constants (in the Advanced tab). Click OK, then click Close, to close both dialog boxes.

Turbulence inlet boundary conditions: Click the Apply Load icon , set the Load Type to Turbulence and click the Define… button to the right of the Load Number field. In the Define Turbulence dialog box, add turbulence number 1, set the Load Values field to "Computed", set the "Mean Velocity at Boundary" to 3, the "Dissipation Length Scale" to

0.127 and click Save. Note that the Prescribed Value for Kinetic-Energy is reset to 0.0084375 and the Prescribed Value for Rate of Energy Dissipation is reset to 0.020342. Click OK to close the dialog box.

In the Apply Usual Boundary Conditions/Loads dialog box, set the 'Apply to' field to Surface, set the Surface # to 1 in the first row of the table and click OK. When you click the Load Plot icon , the graphics window should look something like this:



Turbulence initial conditions: Choose Model→Initial Conditions→Define…, add Condition Name I1, edit the table to read

Variable	Value
K-ENERGY	0.0084375
E-DISSIPATION	0.020342

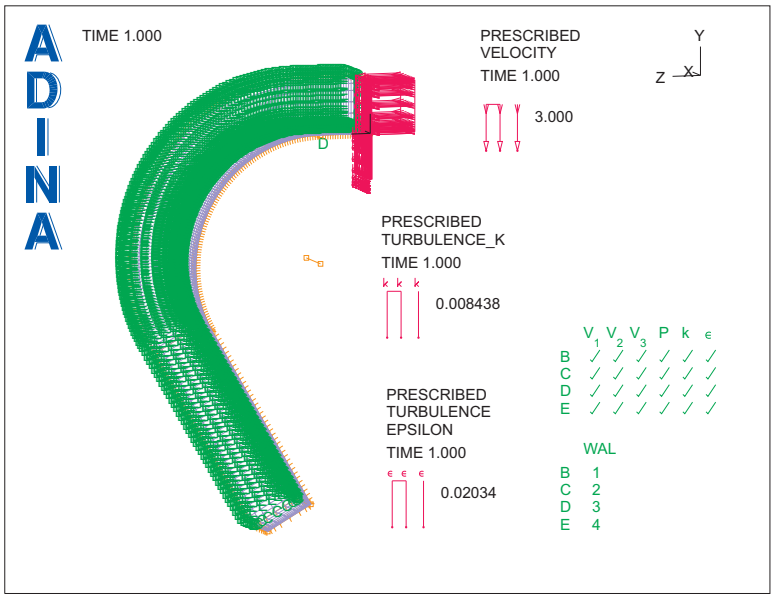
and click Save (these values are same values as are applied at the inlet). Now click the Apply… button, set the "Apply to" field to Volume, set the Volume # to 1, 2, 3 in the first three rows in the table, then click OK twice to close both dialog boxes.

Defining the rest of the model

We have prepared a batch file (`prob42_2.in`) that contains the rest of the model definitions:

- ► Prescribed velocity boundary condition at inlet
- ► Subdivision data
- ► Element group data
- ► Meshing

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob42_2.in` and click Open. The graphics window should look something like this:



Generating the data file, running ADINA-CFD, loading the porthole file

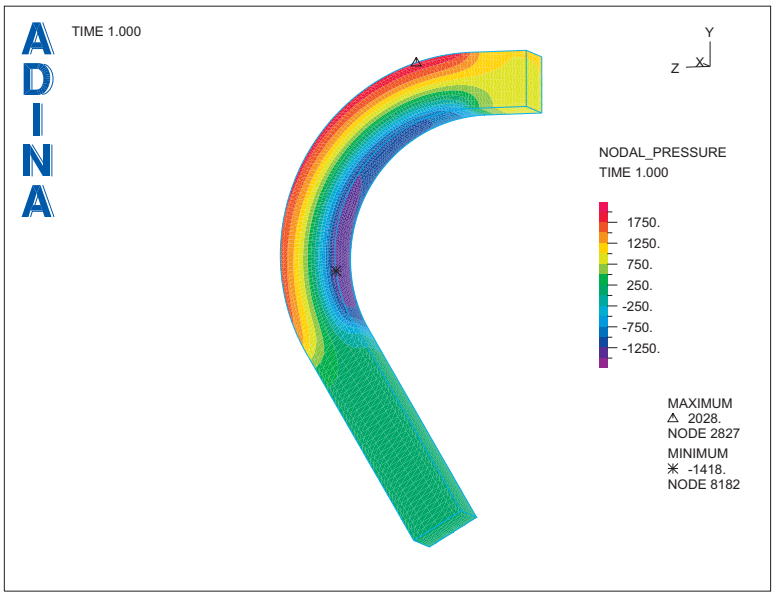
Click the Save icon and save the database to file `prob42`. Click the Data File/Solution icon, set the file name to `prob42`, make sure that the Run Solution button is checked, make sure that 'Maximum Memory for Solution' is at least 30 MB and click Save.

When ADINA-CFD is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon, set the "File type" field to "ADINA-IN Database Files (*.idb)", open database file `prob42`, click the Open icon and open porthole file `prob42`.

Please notice that we first opened the ADINA-IN database file, then loaded the porthole file.

Post-processing

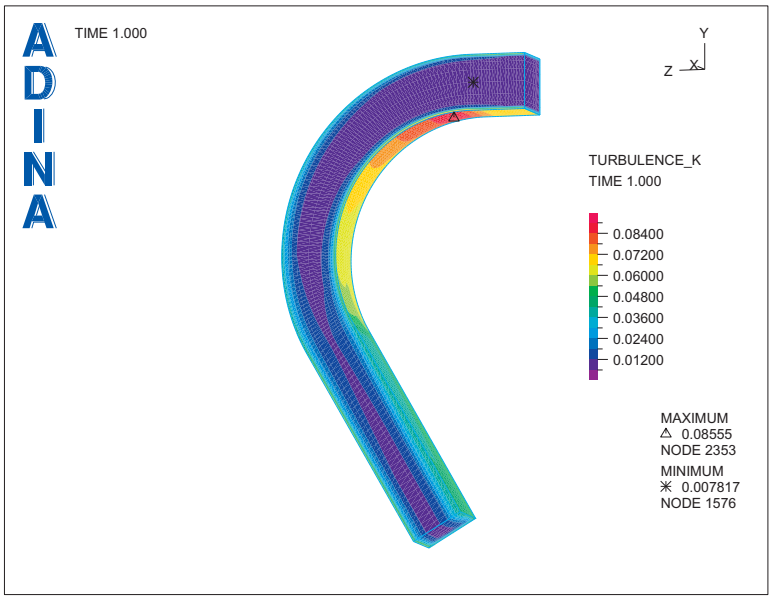
Pressure distribution: Click the Clear icon, the Model Outline icon, the Show Geometry icon (to hide the geometry), and the Quick Band Plot icon. The graphics window should look something like this:

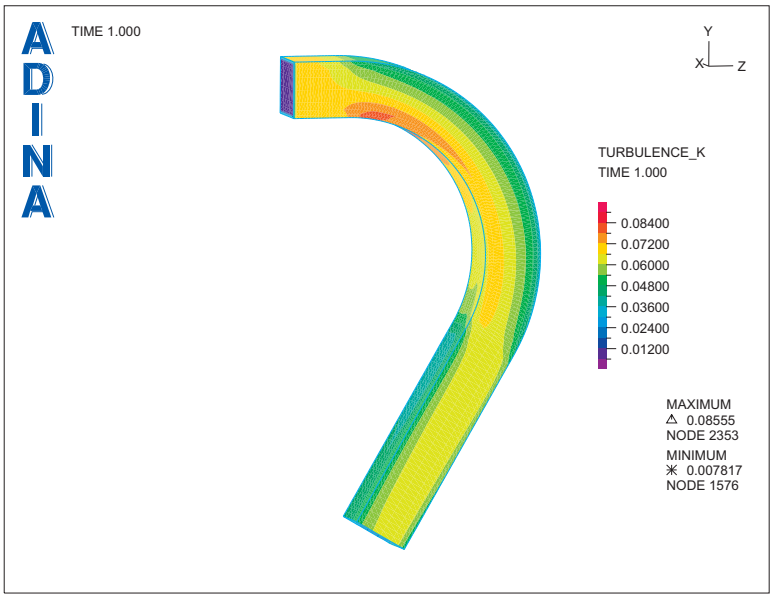


The pressure is highest at the outer radius and lowest at the inner radius.

Turbulence distribution: Click the Modify Band Plot icon, set the Variable to (Fluid Variable: TURBULENCE_K) and click OK. The graphics window should look something like the top figure on the next page.

Use the Pick icon and the mouse to rotate the mesh plot until the graphics window looks something like the bottom figure on the next page. Notice that the turbulence is highest on the no-slip walls, as expected.


ADINA
TIME 1.000
Y
Z
X
TURBULENCE_K
TIME 1.000
0.08400
0.07200
0.06000
0.04800
0.03600
0.02400
0.01200
MAXIMUM
0.08555
NODE 2353
MINIMUM
0.007817
NODE 1576


ADINA
TIME 1.000
Y
X
Z
TURBULENCE_K
TIME 1.000
0.08400
0.07200
0.06000
0.04800
0.03600
0.02400
0.01200
MAXIMUM
0.08555
NODE 2353
MINIMUM
0.007817
NODE 1576

Static pressure coefficient graph

We would like to plot the static pressure coefficient as a function of the angular position around the channel, along the line of symmetry for both the outer radius and inner radius. The static pressure coefficient is defined as $C_p = \frac{p - p_0}{\frac{1}{2}\rho V^2}$ where V is the inlet velocity and p_0 is a reference pressure, chosen so that $C_p = -1$ at the inlet. Since it is more convenient to work with the inlet pressure as the reference pressure, we write $C_p = \frac{p - p_{inlet}}{\frac{1}{2}\rho V^2} - 1$.

From Khalil and Weber, Figure 6, the static pressure coefficient at the outlet is about –1.12.

First we need to determine the inlet pressure. We will use the average of the two pressures at the inlet sampled at the outer radius and at the inner radius (both samples at the line of symmetry). Choose Definitions→Model Point (Combination)→General, add point INLET_AVERAGED, set the Type to Average, enter

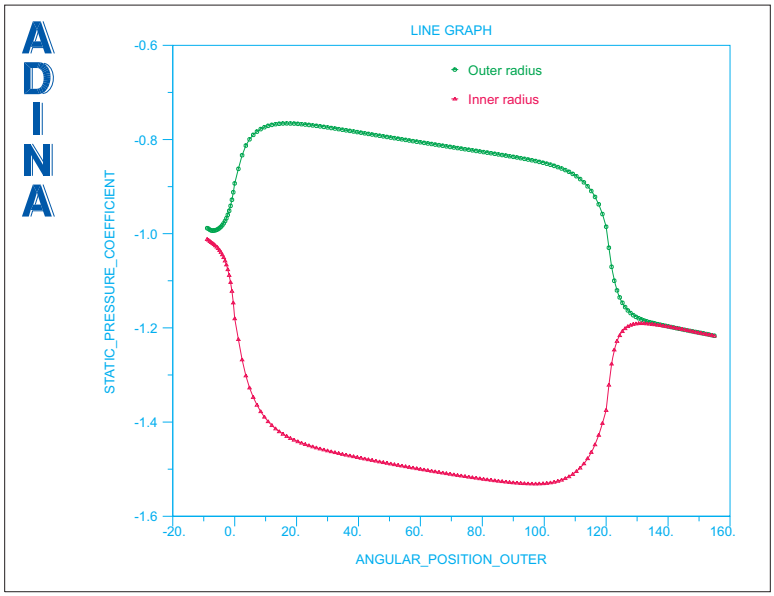
POINT 2
POINT 3

in the first two rows of the table, then click OK. Now choose List→Value List→Model Point, make sure that the Model Point Name is INLET_AVERAGED, set the Variable to (Stress: NODAL_PRESSURE) and click Apply. The result is 9.74715E+02. Click Close to close the dialog box.

Now we create the static pressure coefficent graph. We have prepared the commands for creating the static pressure coefficient graph in file `prob42_1.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob42_1.plo` and click Open. The AUI processes the commands in the batch file. The graphics window should look something like the figure on the next page.

(Note, part of the command file input is the inlet pressure. To save time, we have already set the inlet pressure in the command file to 9.74715E+02).

Choose Graph→List and scroll to the bottom of the dialog box. The static pressure coefficient for the last point is –1.21660. Click Close to close the dialog box.



Restart to second order FCBI-C elements

By default, the FCBI-C elements are first order in space. We can improve the solution by using second order FCBI-C elements.

Set the Program Module drop-down list to ADINA CFD (you can discard all changes) and choose file `prob42.idb` from the recent file list near the bottom of the File menu.

Choose Control→Solution Process and click the Restart Analysis button. Now click the Outer Iteration… button, click the Advanced Settings… button, set the 'Space Discretization Accuracy Order' to Second and click OK three times to close all three dialog boxes.

Generating the data file, running ADINA-CFD, loading the porthole file

Click the Save icon to save the database file. Click the Data File/Solution icon, set the file name to `prob42b`, make sure that the Run Solution button is checked, make sure that 'Maximum Memory for Solution' is at least 30 MB and click Save. The AUI opens a window in which you can specify the restart file from the first analysis. Enter restart file `prob42` and click Copy.

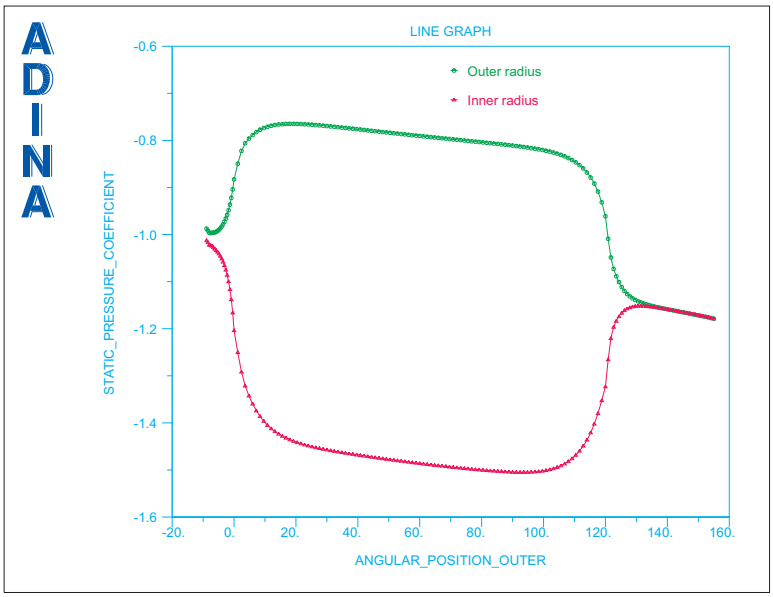
Notice that ADINA-CFD takes less time to run. This is because the initial conditions (from the first order solution) are close to the second order solution.

When ADINA-CFD is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon, set the "File type" field to "ADINA-IN Database Files (*.idb)", open database file `prob42`, click the Open icon and open porthole file `prob42b`.

Post-processing

Static pressure coefficient graph

We have prepared the commands for creating the static pressure coefficient graph in file `prob42_2.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob42_2.plo` and click Open. The AUI processes the commands in the batch file. The graphics window should look something like this:



Choose Graph→List and scroll to the bottom of the dialog box. The static pressure coefficient for the last point is –1.17858, which is quite comparable to the value from Khalil and Weber. Click Close to close the dialog box.

Mass flux calculations: We would like to calculate the mass flux into and out of the channel. To do this, we need element face-sets corresponding to the channel inlet and outlet.

Click the Clear icon and the Mesh Plot icon.

ADINA-CFD provides an element face-set for the channel inlet. Click the Element Face Set icon and select face-set 5. Move the dialog box out of the way of the mesh plot and notice that the channel inlet is highlighted. Click Cancel to close the dialog box.

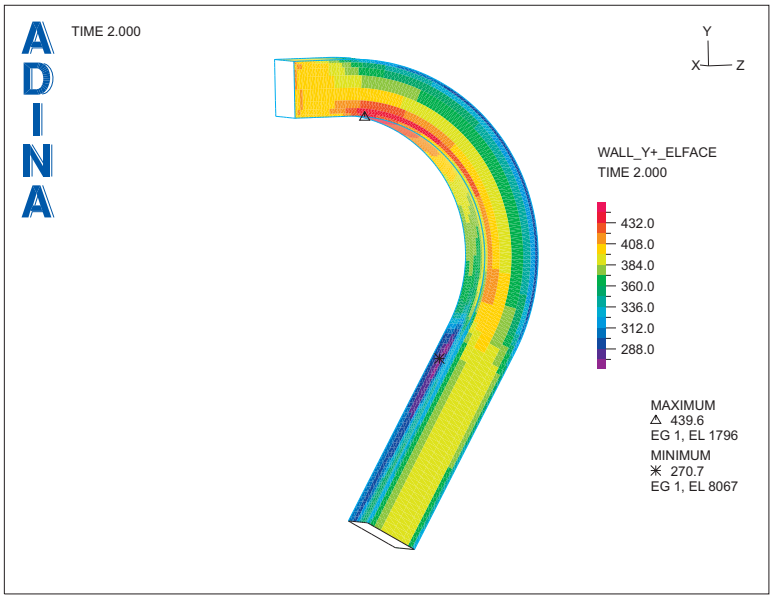
Choose Definitions→Model Point→Element Face Set, add name INLET, set the Element Face Set # to 5 and click OK. Now choose List→Value List→Model Point, set Variable 1 to (Flux: MASS_FLUX_ELFACE) and click Apply. The mass flux should be 4.83870E+01. This mass flux is the same as the value obtained from ρVA where $A = H^2$. Click Close to close the dialog box.

Now let's determine the mass flux at the channel outlet. ADINA-CFD does not provide an element face-set for the channel outlet because there is no boundary condition defined at the outlet. Click the Element Face Set icon and add face-set 10, set the Method to "From Surfaces/Faces", set the Surface/Face # to 16 in the first row of the table and click Save. Move the dialog box out of the way of the mesh plot and notice that the channel outlet is highlighted. Click OK to close the dialog box.

Choose Definitions→Model Point→Element Face Set, add name OUTLET, set the Element Face Set # to 10 and click OK. Now choose List→Value List→Model Point, set the Model Point Name to OUTLET, set Variable 1 to (Flux: MASS_FLUX_ELFACE) and click Apply. The mass flux should be -4.83870E+01. It is seen that the sum of the mass fluxes is zero, so that mass is conserved.

Plotting the dimensionless wall distance: We would like to plot the dimensionless wall distance. Click the Clear icon and the Model Outline icon, then click the Show Geometry icon (to hide the geometry). Now click the Create Band Plot icon, set the Band Plot Variable to (Fluid Variable: WALL_Y+_ELFACE) and click OK. Use the mouse to rotate the mesh so that the graphics window looks something like the figure on the next page.

Exiting the AUI: Choose File→Exit (you can discard all changes).


ADINA
TIME 2.000
Y
X
Z
WALL_Y+_ELFACE
TIME 2.000
432.0
408.0
384.0
360.0
336.0
312.0
288.0
MAXIMUM
439.6
EG 1, EL 1796
MINIMUM
270.7
EG 1, EL 8067