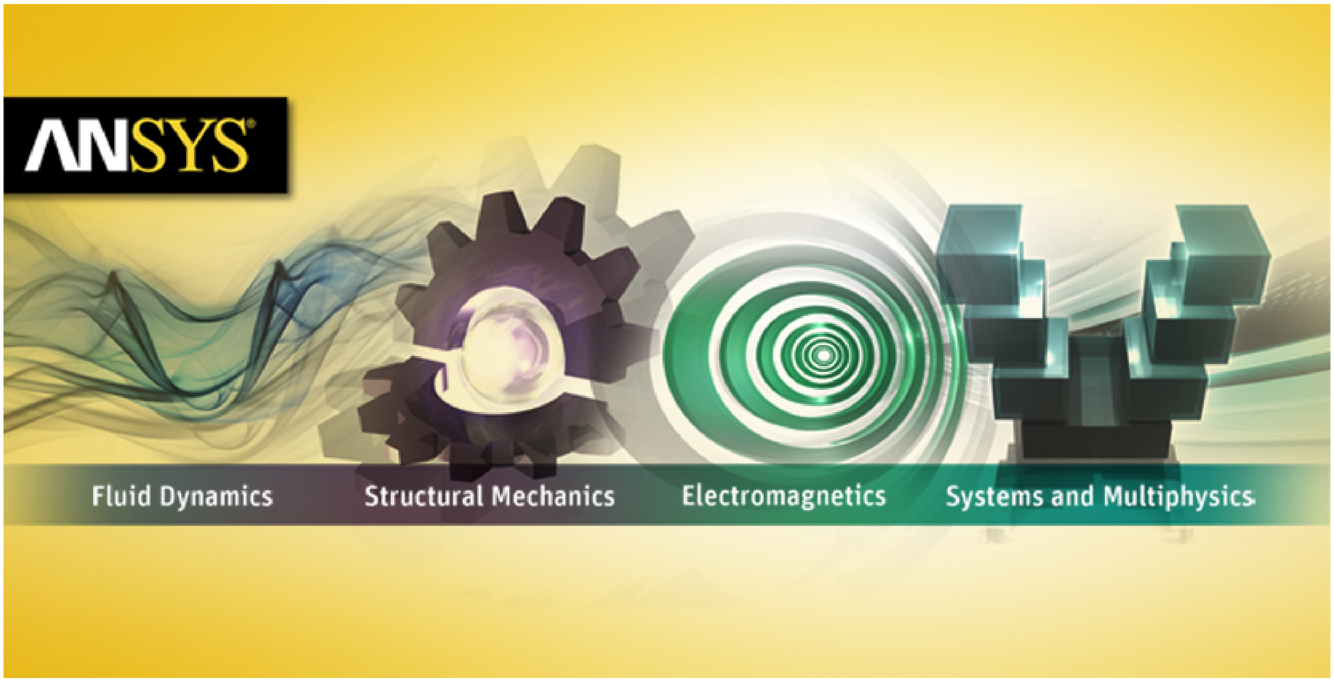


POLYFLOW in Workbench Tutorial

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 14.0
November 2011

ANSYS, Inc. is certified to ISO 9001:2008.

Published in the U.S.A.

Table of Contents

Chapter 1: ANSYS POLYFLOW in ANSYS Workbench Tutorial: 3D Extrusion

1.1. Introduction

This tutorial illustrates how to use ANSYS POLYFLOW fluid flow systems in ANSYS Workbench to set up and solve a 3D extrusion problem with a variety of inlet flow rates. This tutorial is designed to introduce you to the ANSYS Workbench tool set using the same geometry that is used in Tutorial 6 in the separate Tutorial Guide. In this tutorial, you will import the geometry and generate a computational mesh using the geometry and meshing tools within ANSYS Workbench. Then you will use ANSYS POLYDATA to modify an imported data file, solve the CFD problem using ANSYS POLYFLOW, and view the results in the CFD-Post postprocessing tool. Finally, you will use the **Parameter and Design Points** view in ANSYS Workbench to calculate results for multiple design points that represent different inlet flow rates.

This tutorial demonstrates how to do the following:

- Launch ANSYS Workbench.
- Create an ANSYS POLYFLOW fluid flow analysis system in ANSYS Workbench.
- Import and edit geometry using ANSYS DesignModeler.
- Create a computational mesh for the geometry using the ANSYS Meshing application.
- Import a data file, and modify it using ANSYS POLYDATA to include a user-defined template for the die inlet flow rate.
- Calculate a solution using ANSYS POLYFLOW.
- View the initial results and create an output parameter for the maximum velocity of the extrudate in CFD-Post.
- Generate results for multiple design points using the **Parameter and Design Points** view, and chart how the outflow velocity varies with the inlet flow rate.

1.2. Prerequisites

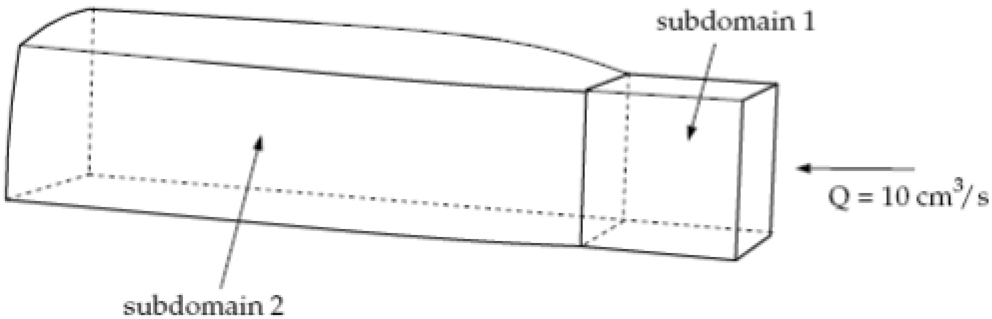
This tutorial assumes that you have little to no experience with ANSYS DesignModeler, ANSYS Meshing, ANSYS POLYFLOW, CFD-Post, or the **Parameter and Design Points** view of ANSYS Workbench, and so each step will be explicitly described.

1.3. Problem Description

This problem deals with the flow of a Newtonian fluid through a three-dimensional die. Due to the symmetry of the problem (the cross-section of the die is a square), the computational domain is defined for a quarter of the geometry and two planes of symmetry are defined.

The melt enters the die as shown in *Figure 1.1* (p. 2) at an initial flow rate of $Q = 10$ cm^3/s (this flow rate is a quarter of that for the complete physical system) and the extrudate is obtained at the exit. It is assumed that the extrudate is fully deformed at the end of the computational domain, and that it will not deform any further (i.e., subdomain 2 is long enough to account for all the deformation of the extrudate).

Figure 1.1 Problem Description

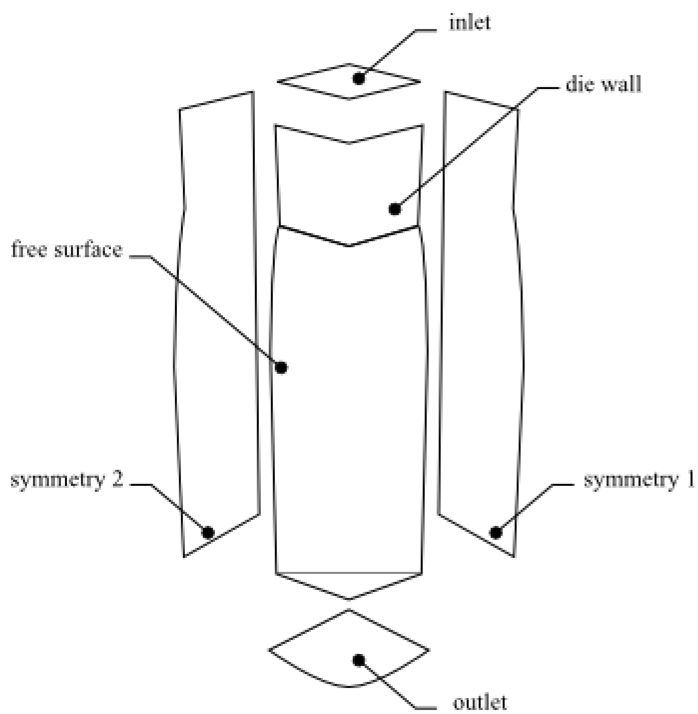


The incompressibility and momentum equations are solved over the computational domain. The domain for the problem is divided into two subdomains (as shown in *Figure 1.1* (p. 2)) so that a remeshing algorithm can be applied only to the portion of the mesh that will be deformed. Subdomain 1 represents the die where the fluid is confined. Subdomain 2 corresponds to the extrudate that is in contact with the air and can deform freely. The calculation will determine the location of the free surface (the skin of the extrudate), as well as the velocity of the extrudate at the exit.

The boundary set for the problem is shown in *Figure 1.2* (p. 3), and the conditions at the boundaries of the domains are:

- inlet: flow inlet, initial volumetric flow rate $Q = 10$ cm^3/s
- die wall: zero velocity
- free surface: free surface
- symmetry 1: symmetry plane
- symmetry 2: symmetry plane
- outlet: flow exit

Figure 1.2 The Boundary Set for the Problem



1.4. Setup and Solution

1.4.1. Preparation

1. Copy the file `ext3d-workbench.zip` to your working directory. To access this file, begin by pointing your web browser to

   - For Windows:

     path`\ANSYS Inc\v140\polyflow\polyflow14.0.`*x*`\help\index.htm`

   - For Linux:

     path`/ansys_inc/v140/polyflow/polyflow14.0.`*x*`/help/index.htm`

   where *path* is the directory in which ANSYS POLYFLOW has been installed and x represents the appropriate number for the release (e.g., `0` for `polyflow14.0.0`).

   If, for example, you are using Internet Explorer as your browser, select the **File** > **Open...** menu item and click the **Browse** button to browse through your directories to find the file.

   When opened, the file displays the ANSYS POLYFLOW documentation "home" page. Click the **Download** link under the **ANSYS POLYFLOW in ANSYS Workbench Tutorial** heading, and then copy the `ext3d-workbench.zip` file that is saved to your computer to your working directory.

2. Unzip `ext3d-workbench.zip`.

The geometry file `ext3d.x_t` *and the data file* `polyflow.dat` *can be found in the* `ext3d-workbench` *folder created after unzipping the file. Solution files created during the preparation of the tutorial are provided in a* `solution_files` *folder.*

Note

This tutorial is prepared using ANSYS POLYFLOW on a Windows system. The screen shots and graphic images that follow may be slightly different than the appearance on your system, depending on the operating system or graphics card.

1.4.2. Step 1: Creating a Fluid Flow Analysis System in ANSYS Workbench

In this step, you will start ANSYS Workbench, create a new fluid flow analysis system, then view the list of files generated by ANSYS Workbench.

1. Start ANSYS Workbench by clicking the **Start** button, moving your pointer over **All Programs**, moving your pointer over **ANSYS 14.0**, then clicking **Workbench 14.0**.

   Start → **All Programs** → **ANSYS 14.0** → **Workbench 14.0**

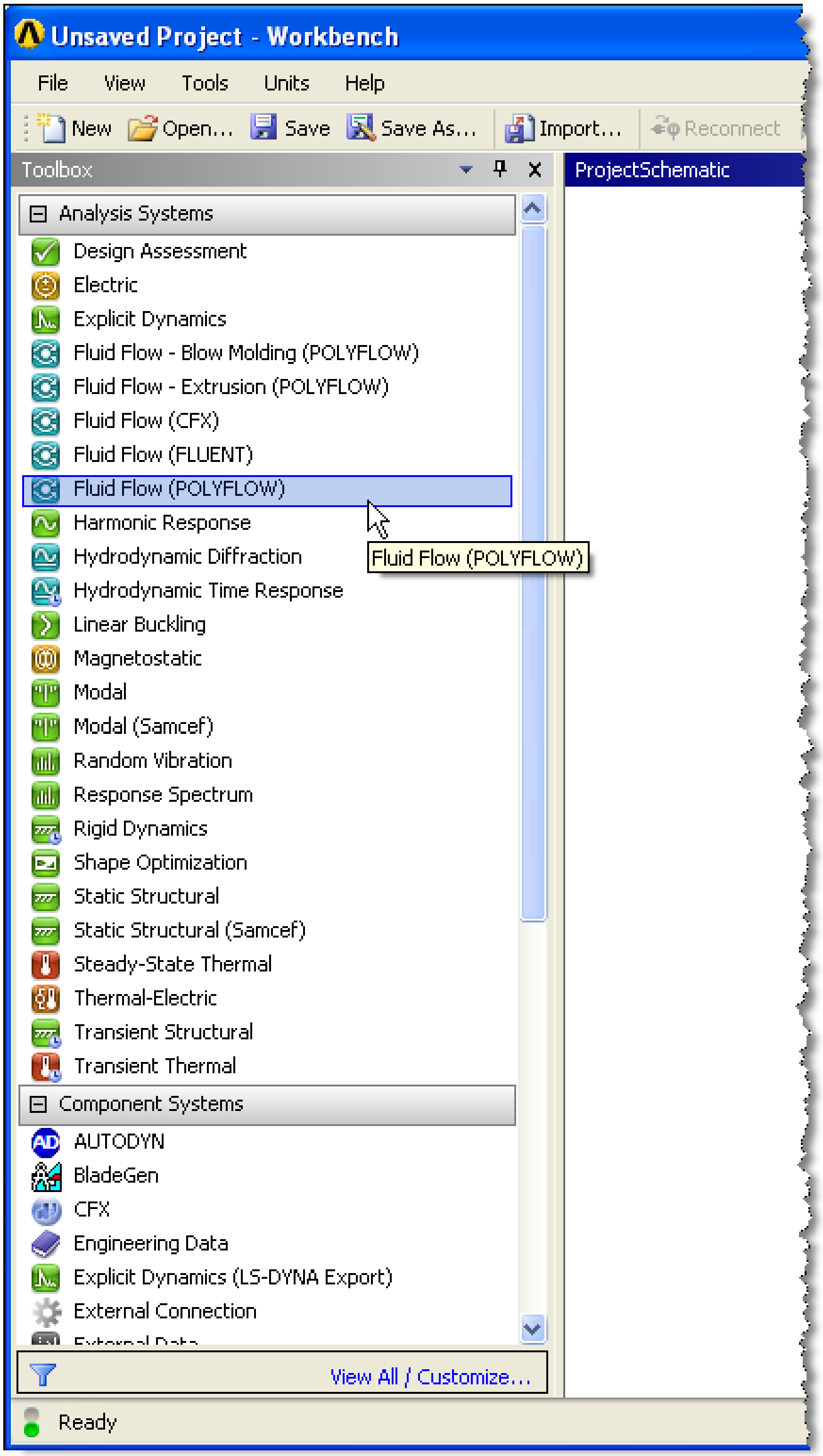
   The ANSYS Workbench application window will open, containing the **Toolbox** *on the left and the* **Project Schematic** *on the right. Various supported analyses and applications are listed in the* **Toolbox**, *while you visualize the components of the analysis in the* **Project Schematic**.

   Note

   When you first start ANSYS Workbench, the **Getting Started** pop-up window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking the red 'X' icon in the upper right corner. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

2. Create a new fluid flow analysis system by double-clicking the **Fluid Flow (POLYFLOW)** option under **Analysis Systems** in the **Toolbox**.

Figure 1.3 Selecting the Fluid Flow (POLYFLOW) Analysis System in ANSYS Workbench

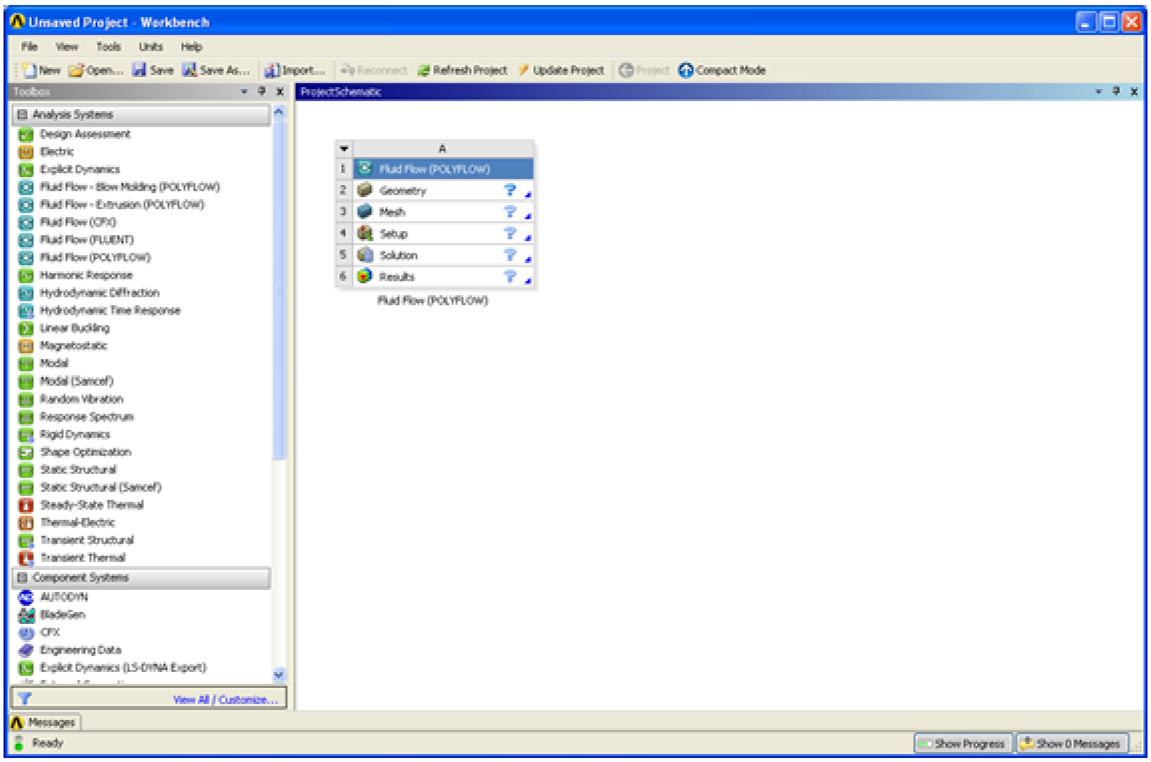


Extra

You can also create a new fluid flow analysis system by dragging-and-dropping the analysis system into the Project Schematic: a green dotted outline will indicate a potential location in the Project Schematic for the new system, which will turn into a red box when you attempt to drop it.

A new ANSYS POLYFLOW-based fluid flow analysis system will be displayed in the **Project Schematic**.

Figure 1.4 ANSYS Workbench with a New ANSYS POLYFLOW-Based Fluid Flow Analysis System



Note

The ANSYS POLYFLOW-based fluid flow analysis system, for example, is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the work flow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated (e.g., ANSYS POLYFLOW) and native applications into a single, seamless project flow, where individual cells can obtain data from and provide data to other cells. ANSYS Workbench provides visual indications of a cell's *state* at any given time via icons on the right side of each cell. Because of the constant flow of data, a cell's state can quickly change. Brief descriptions of the various states are provided below. For more information about cell states, see the ANSYS Workbench online help.

- **Unfulfilled** ( ) indicates that required upstream data does not exist. For example, when you first create a new **Fluid Flow (POLYFLOW)** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

- **Refresh Required** ( ) indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the **Geometry** cell in your new **Fluid Flow (POLYFLOW)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- **Attention Required** ( ) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch ANSYS POLY-DATA from the **Setup** cell in a **Fluid Flow (POLYFLOW)** analysis system that has a valid mesh, the **Setup** cell appears as **Attention Required** because additional data must be entered in ANSYS POLYDATA before you can calculate a solution.

- **Update Required** ( ) indicates that local data has changed and the output of the cell needs to be regenerated. For example, after you launch ANSYS Meshing from the **Mesh** cell in a **Fluid Flow (POLYFLOW)** analysis system that has a valid geometry, the **Mesh** cell appears as **Update Required** because the **Mesh** cell has all the data it needs to generate an ANSYS POLYFLOW mesh file, but the ANSYS POLYFLOW mesh file has not yet been generated.

- **Up-to-Date** ( ) indicates that an update has been performed on the cell and no failures have occurred (or an interactive calculation has been completed successfully). For example, after ANSYS POLYFLOW finishes performing the number of necessary solver iterations, the **Solution** cell appears as **Up-to-Date**.

- **Interrupted** ( ) indicates that you have interrupted an update (or stopped an interactive calculation that is in progress). For example, if you select the stop button ( ) in the Progress Monitor of ANSYS Workbench at a point where ANSYS POLYFLOW has generated results but has not yet completed the calculation (such as during a transient simulation), then verify the action in the dialog box that opens, ANSYS POLYFLOW is immediately stopped and the **Solution** cell appears as **Interrupted**.

- **Input Changes Pending** ( ) indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date Fluid Flow (POLYFLOW)** analysis system, the **Setup** cell appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

- **Pending** ( ) indicates that a batch or asynchronous solution is in progress. This icon will only appear when the **Solution** cell is in background mode.

- **Refresh Failed, Refresh Required** ( ) indicates that the last attempt to refresh cell input data failed, and so the cell needs to be refreshed.

- **Update Failed, Update Required** ( ) indicates that the last attempt to update the cell and calculate output data failed, and so the cell needs to be updated. For example, if you update the **Solution** cell and the solver diverges during the calculation, the **Solution** cell appears as **Update Failed, Update Required**.

- **Update Failed, Attention Required** ( ) indicates that the last attempt to update the cell and calculate output data failed, and so the cell requires attention.

3. Name the analysis.

   a. Double-click the **Fluid Flow (POLYFLOW)** label underneath the analysis system.

   b. Enter `ext3d` for the name of the analysis system.

4. Save the project.

   a. Select the **Save** option under the **File** menu in ANSYS Workbench.

File → **Save**

*The **Save As** dialog will open, where you can browse to a specific directory and enter a specific name for the ANSYS Workbench project.*

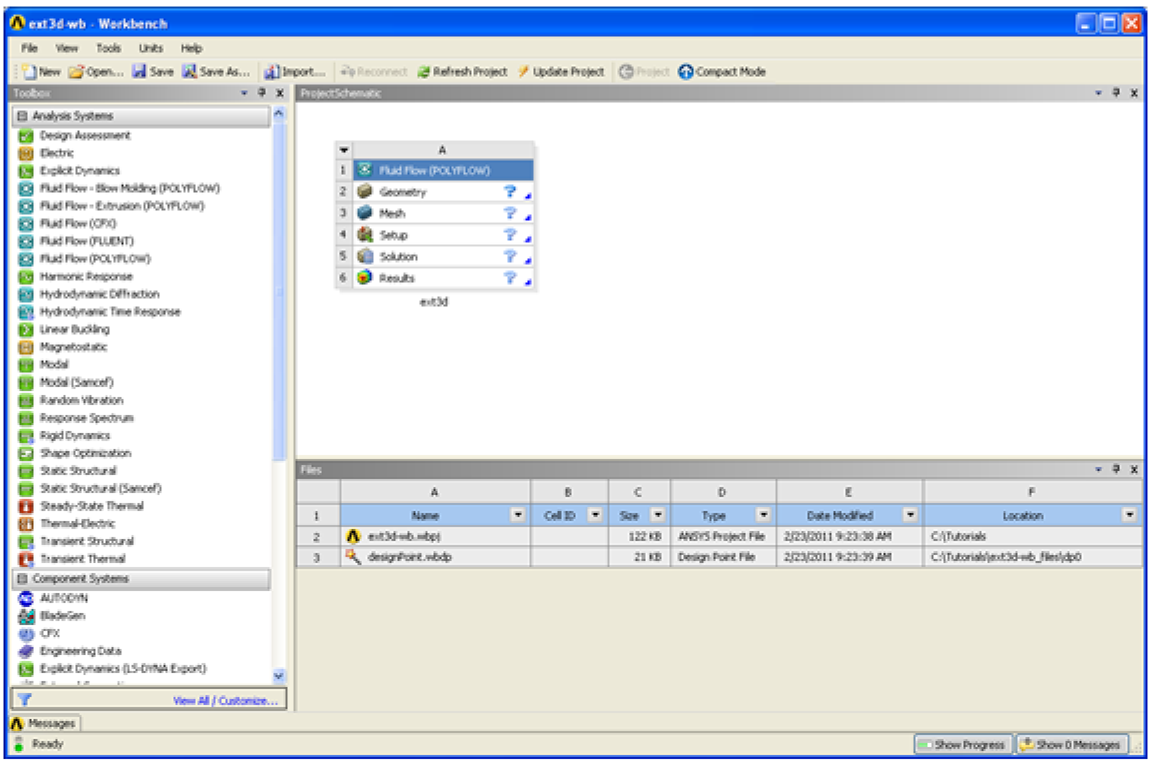
b. In your working directory, enter `ext3d-wb` as the project **File name** and click the **Save** button to save the project. ANSYS Workbench saves the project with a `.wbpj` extension, as well as supporting files for the project.

5. View the files generated by ANSYS Workbench, by enabling the **Files** option under the **View** menu.

   View → **Files**

   *The **Files** view will be displayed in the **Project Schematic**.*

Figure 1.5 Displaying the Files View after Adding an ANSYS POLYFLOW-Based Fluid Flow Analysis System



*ANSYS Workbench allows you to easily view the files associated with your project using the **Files** view. You can see the name and type of file, the ID of the cell the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see the separate **ANSYS POLYFLOW** in **Workbench** User's Guide and the ANSYS Workbench online help.*

1.4.3. Step 2: Preparing the Geometry in ANSYS DesignModeler

In this step, you will import a previously created geometry file, modify the geometry with ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

Note

ANSYS DesignModeler is licensed separately from ANSYS POLYFLOW. If you do not have access to ANSYS DesignModeler, you can instead import a geometry file that does not need to be modified, as noted in step 1.c.

1. Import the geometry file.

    a. Right-click the **Geometry** cell in the `ext3d` fluid flow analysis system (cell A2 in the ANSYS Workbench **Project Schematic**).

    b. Move your pointer over **Import Geometry** in the context menu that opens, and click **Browse...**.

    c. Use the **Open** dialog box to browse to the `ext3d-workbench` folder you unzipped in a previous step, select `ext3d.x_t`, and click **Open**.

    Note

    If you do not have access to ANSYS DesignModeler, select `PFL.agdb` in the **Open** dialog box instead, then skip to *Step 3: Meshing the Geometry in the ANSYS Meshing Application* (p. 11).

    The state of the **Geometry** *cell becomes* **Up-to-Date***, indicating that there is a geometry now associated with the fluid flow analysis system.*

2. Start ANSYS DesignModeler.

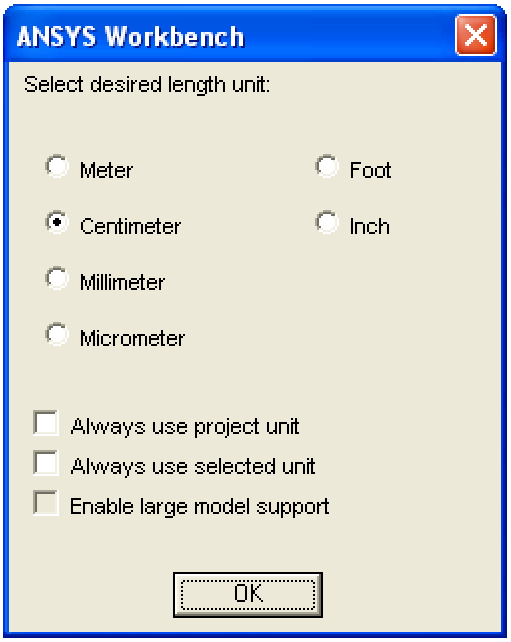
    Double-click the **Geometry** cell in the `ext3d` fluid flow analysis system to launch the ANSYS DesignModeler application.

    Extra

    You can also launch ANSYS DesignModeler by right-clicking on the **Geometry** cell to display the context menu then selecting the **Edit Geometry...** option.

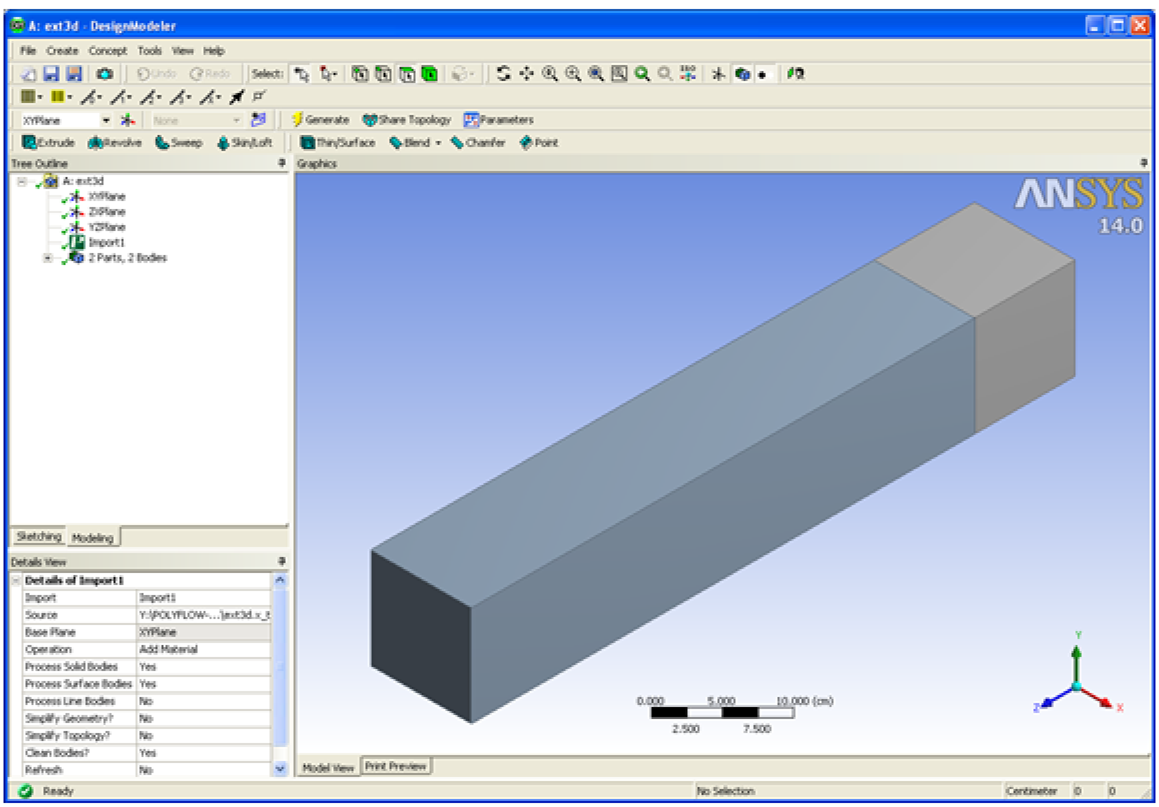
3. Set the units in ANSYS DesignModeler.

    When ANSYS DesignModeler first opens, you are prompted to select the desired system of length units to work from. For the purposes of this tutorial, select **Centimeter** as the desired length unit and click **OK** to close the prompt.



4. Finish importing the geometry file by clicking **Generate** in the ANSYS DesignModeler toolbar. The geometry will be displayed in the **Graphics** window.

Figure 1.6 The Imported Geometry in the ANSYS DesignModeler Application



5. Modify the geometry so that the separate domains ("bodies") are treated as a single entity (a "part"), by performing the following actions in the **Tree Outline**.

   By uniting the multiple bodies of the geometry into a single part, you will create a conformal mesh between the separate domains of the bodies.

a. Expand the **2 Parts, 2 Bodies** node.

b. Click **1** so that it is highlighted.

c. Hold the **Ctrl** key and click **2** so that it is highlighted as well.

d. Right-click the highlighted objects and click **Form New Part** in the menu that opens.

   The **Tree Outline** *will list the geometry as* **1 Part, 2 Bodies**.

6. Close ANSYS DesignModeler.

   You can simply close the ANSYS DesignModeler application. ANSYS Workbench automatically saves the geometry and updates the **Project Schematic** accordingly.

7. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

Files

	A
1	Name
2	ext3d-wb.wbpj
3	ext3d.x_t
4	PFL.agdb
5	designPoint.wbdp

Note the addition of the geometry file (`PFL.agdb`*, where* `PFL` *indicates a POLYFLOW-based fluid flow system) to the list of files.*

1.4.4. Step 3: Meshing the Geometry in the ANSYS Meshing Application

Now that you have prepared the extrusion geometry, you need to generate a computational mesh throughout the flow volume. In this step, you will use the ANSYS Meshing application to create a mesh for your CFD analysis, then review the list of files generated by ANSYS Workbench.

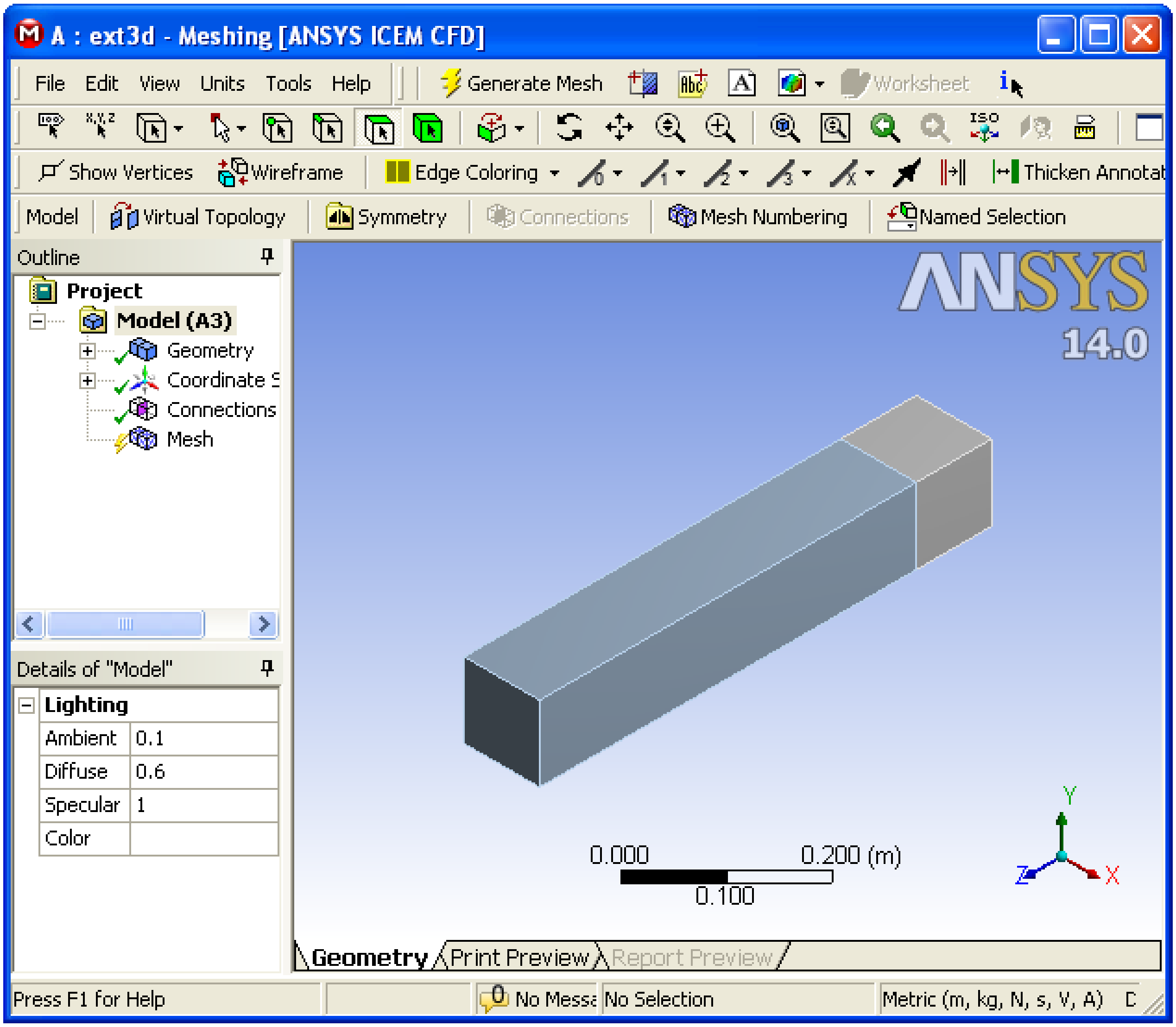
1. Open the ANSYS Meshing application.

   Double-click the **Mesh** cell in the `ext3d` fluid flow analysis system (cell A3) to launch the ANSYS Meshing application with the extrusion geometry already loaded.

   Extra

   You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

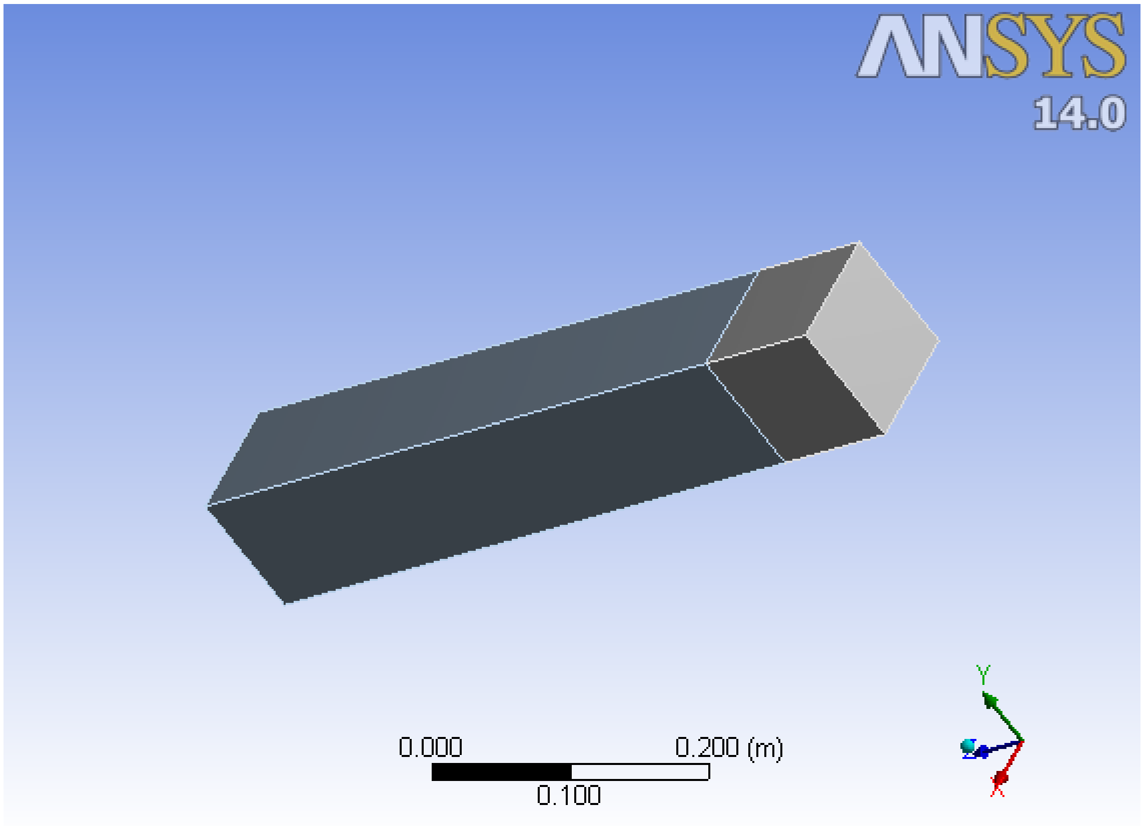
Figure 1.7 The ANSYS Meshing Application with the Extrusion Geometry Loaded



2. Group the faces and create named selections to match the boundary set shown in *Figure 1.2* (p. 3).

   a. Rotate the view to get your display similar to that shown in *Figure 1.8* (p. 13), by holding the center mouse button and moving your pointer in the geometry window.

Figure 1.8 Rotated View



b. Click **Mesh** under **Project/Model** in the **Outline** tree.

Information will be displayed about the mesh in the **Details** *view below the* **Outline** *tree view.*

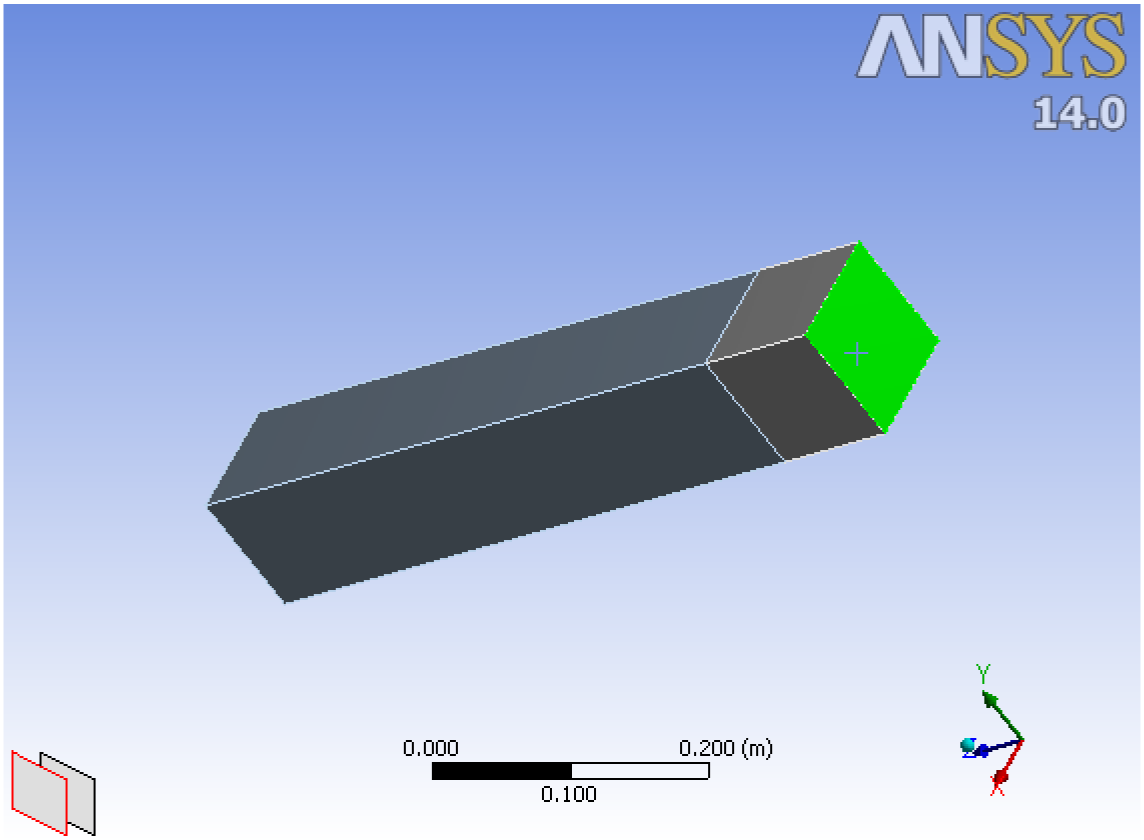
Note

Note that since the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS POLYFLOW, the **Physics Preference** will already be set to **CFD** and the **Solver Preference** will already be set to **POLYFLOW**.

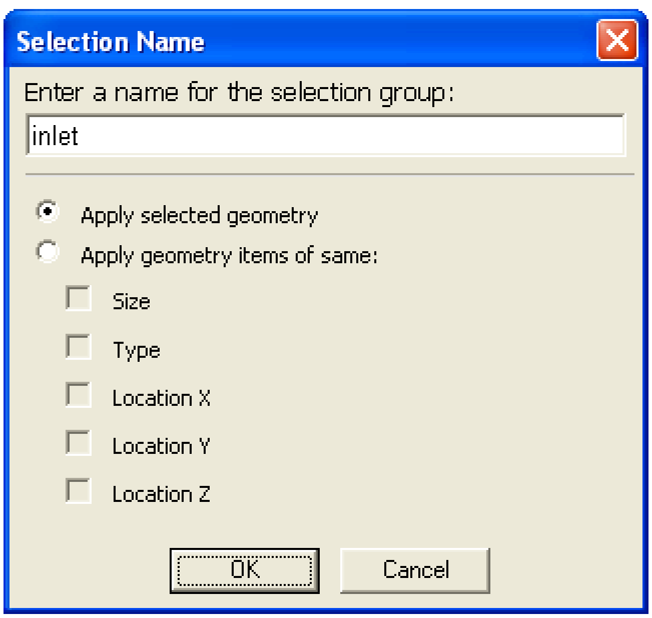
c. Select the face that will represent the inlet, as shown highlighted in green in *Figure 1.9* (p. 14).

d. Right-click and select the **Create Named Selection** option (from the menu that opens) to open the **Selection Name** dialog box.

Figure 1.9 Selecting the Inlet Face

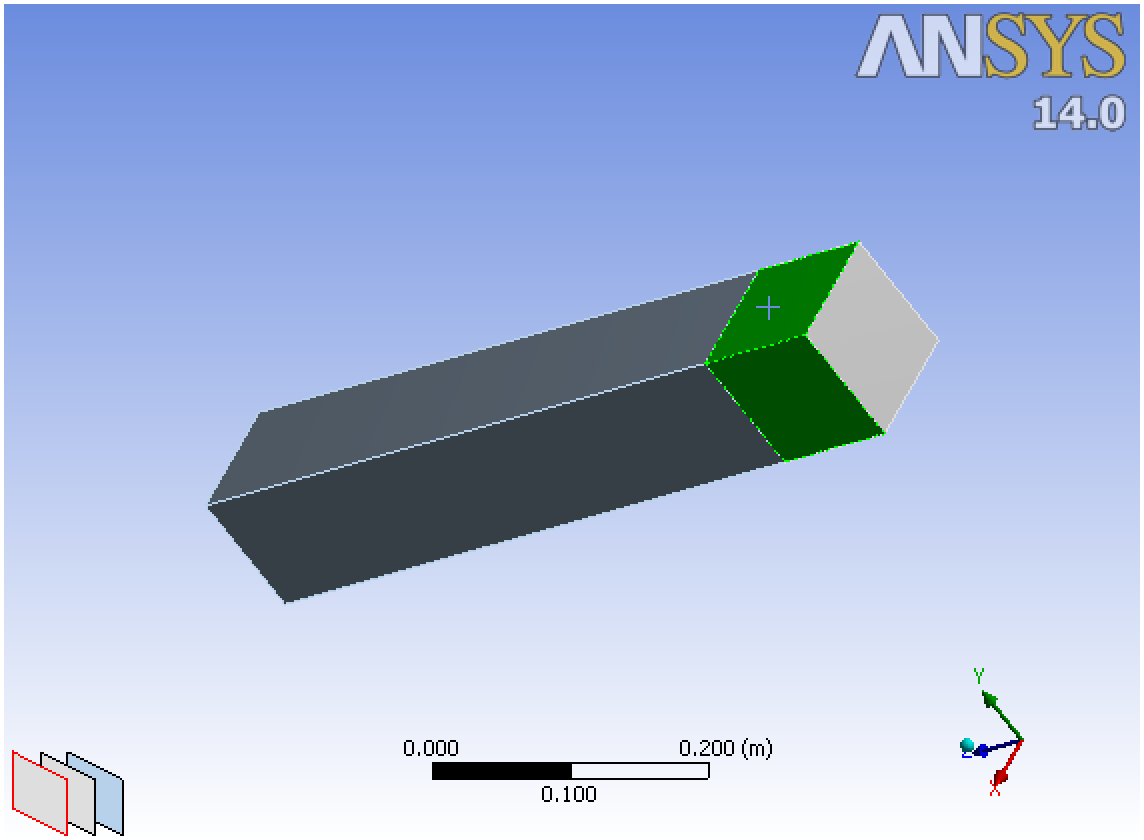


e. Enter `inlet` for the name in the **Selection Name** dialog box, and click **OK**.



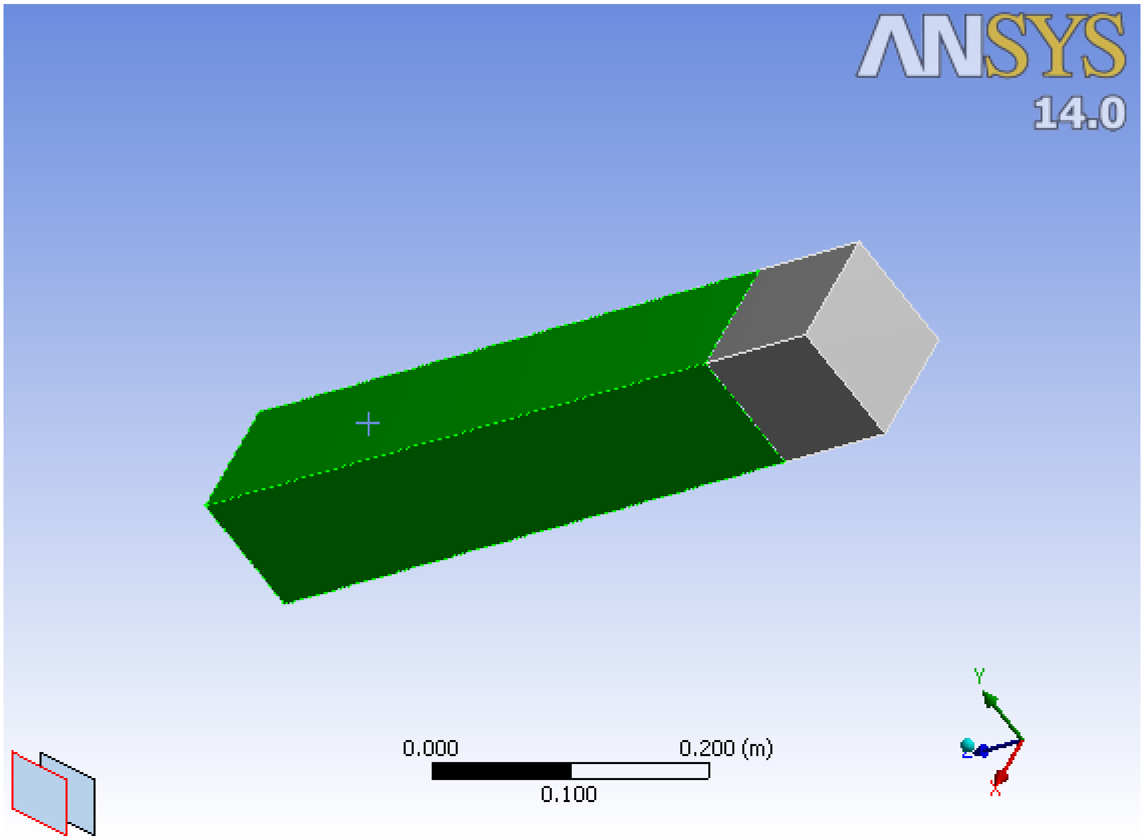
f. Hold down the **Ctrl** key, select the 2 faces that will represent the zero velocity boundary (as highlighted in green in *Figure 1.10* (p. 15)), then create a selection named `die wall` in a manner similar to the previous steps.

Figure 1.10 The Zero Velocity Faces Selected



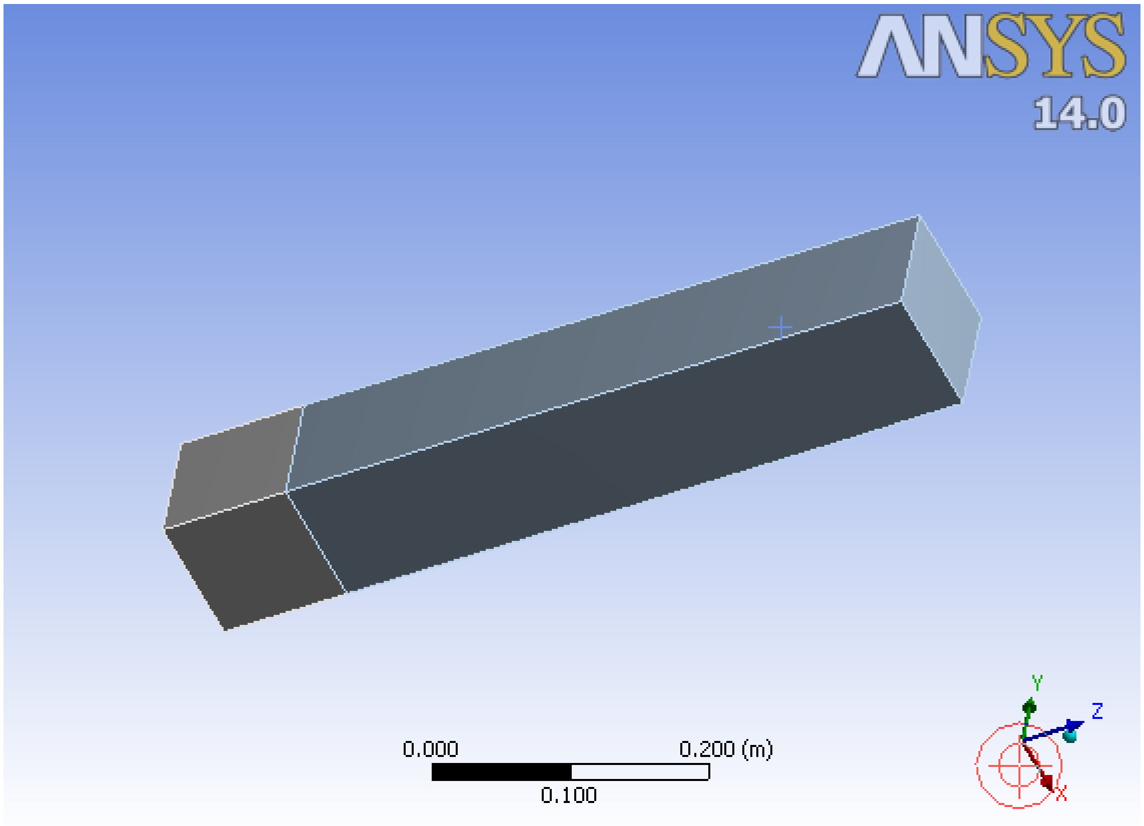
g. Hold down the **Ctrl** key, select the 2 faces that will represent the free surface boundary (as highlighted in green in *Figure 1.11* (p. 16)), and create a selection named `free surface` in a manner similar to the previous steps.

Figure 1.11 The Free Surface Faces Selected



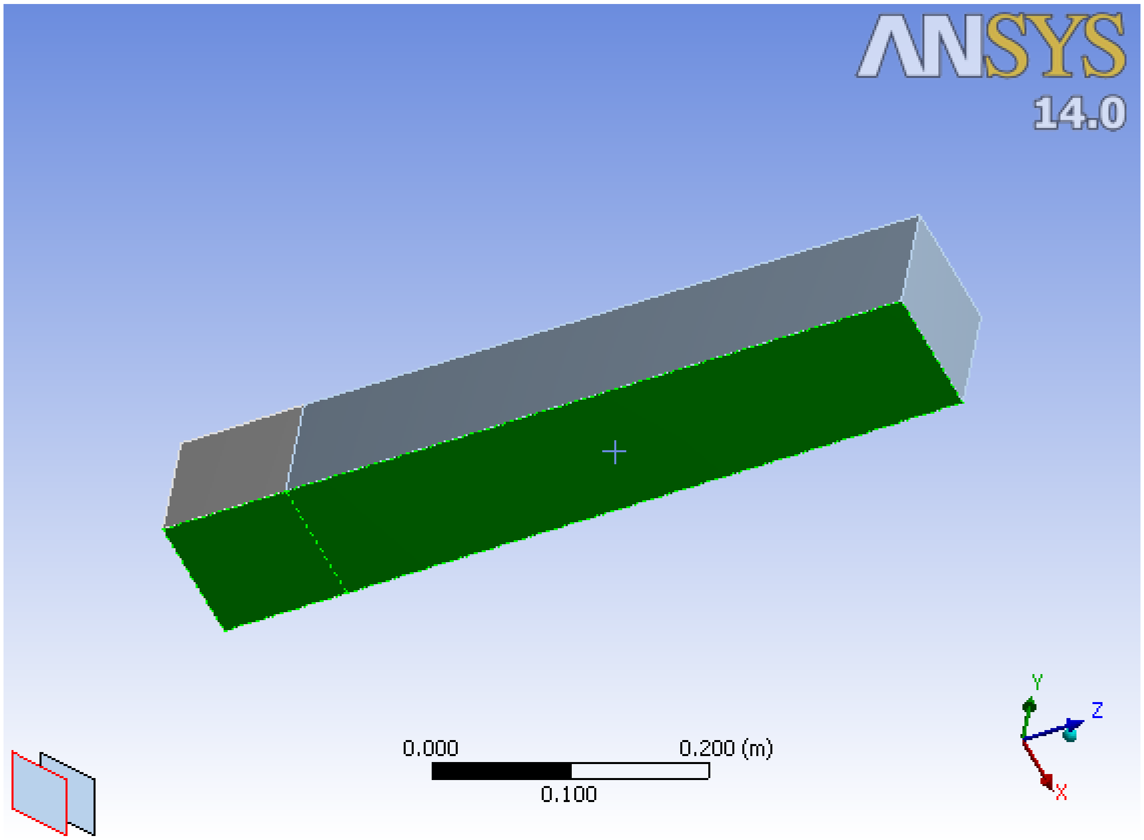
h. Rotate the view to get your display to be similar to that shown in *Figure 1.12* (p. 17), by holding the center mouse button and moving your pointer in the geometry window.

Figure 1.12 Rotated View



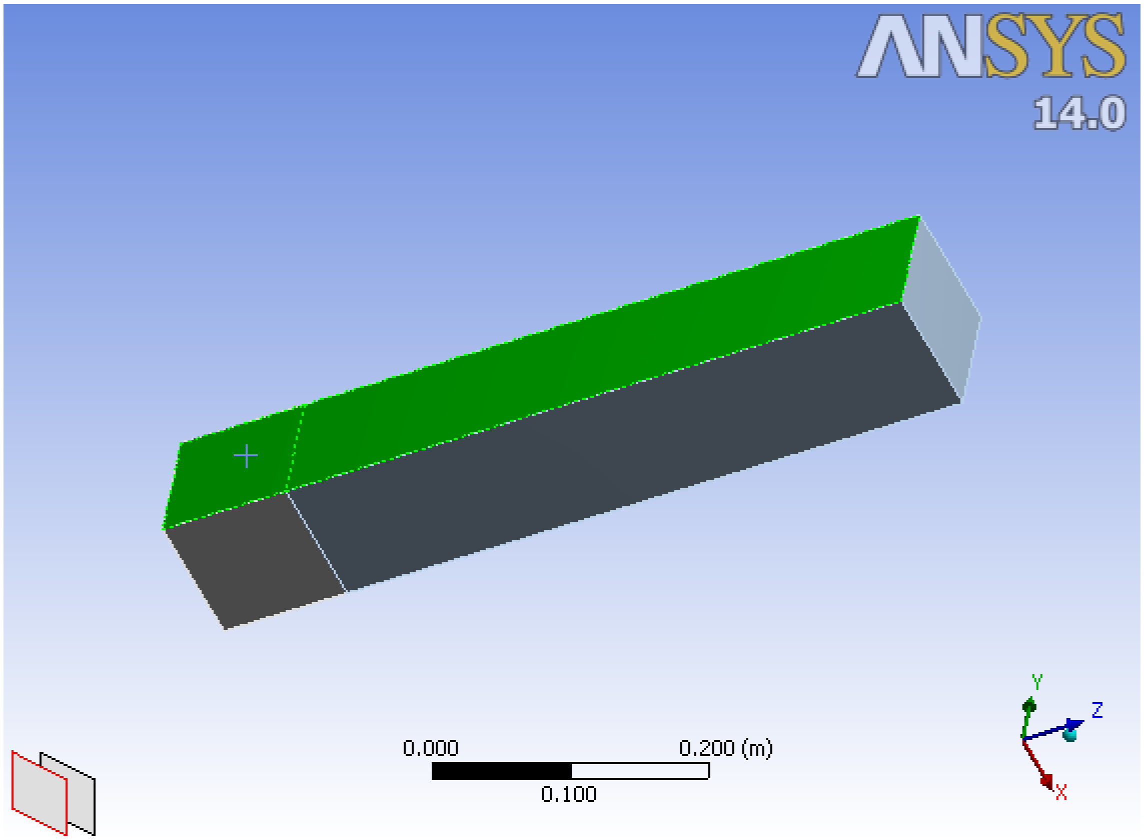
i. Hold down the **Ctrl** key, select the 2 faces that will represent one of the symmetry boundaries (as highlighted in green in *Figure 1.13* (p. 18)), and create a selection named `symmetry 1` in a manner similar to the previous steps.

Figure 1.13 The First Pair of Symmetry Faces Selected



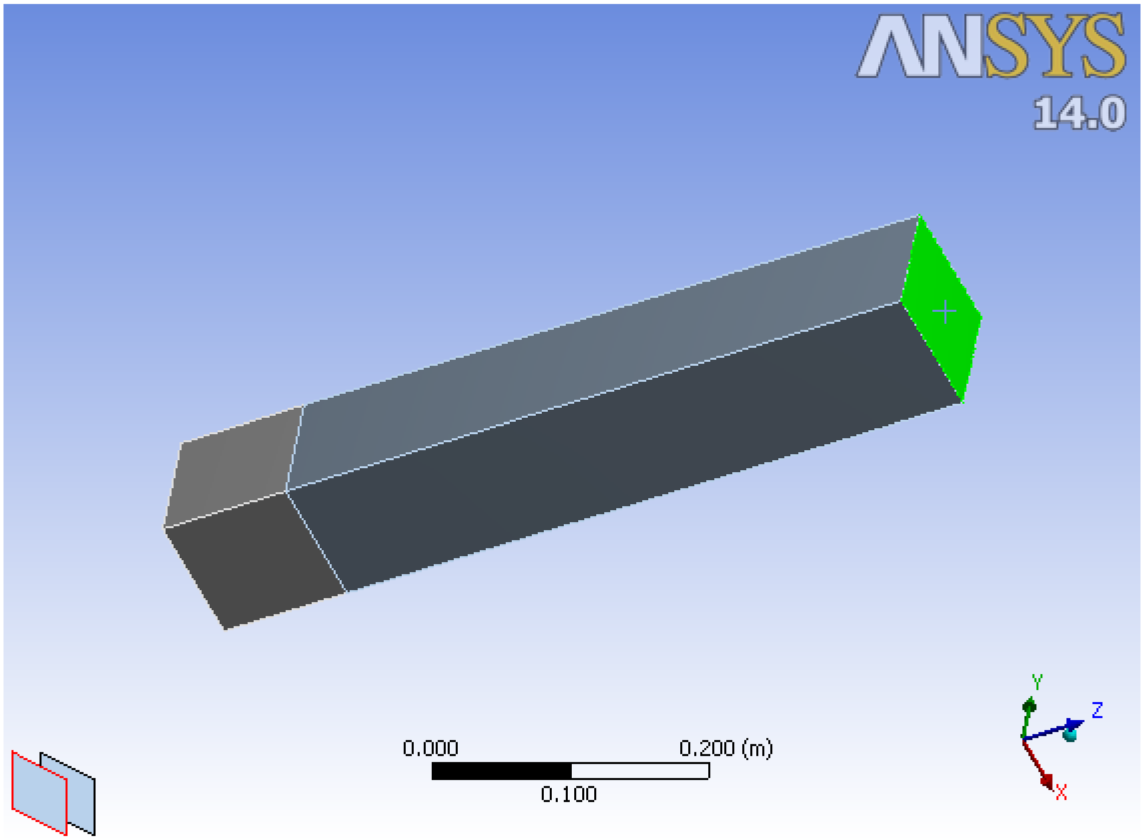
j. Hold down the **Ctrl** key, select the 2 faces that will represent the other of the symmetry boundaries (as highlighted in green in *Figure 1.14* (p. 19)), and create a selection named `symmetry 2` in a manner similar to the previous steps.

Figure 1.14 The Second Pair of Symmetry Faces Selected



k. Select the face that will represent the flow exit boundary (as highlighted in green in *Figure 1.15* (p. 20)), and create a selection named `outlet` in a manner similar to the previous steps.

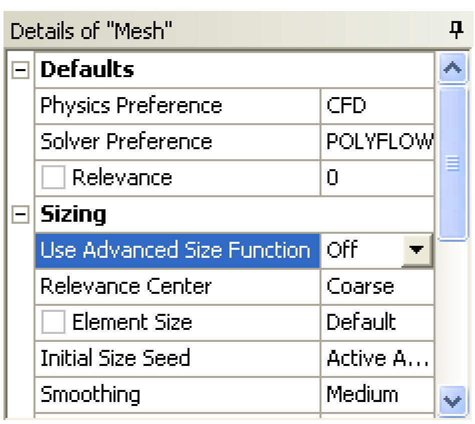
Figure 1.15 The Flow Exit Face Selected



3. Set the appropriate meshing parameters for the ANSYS Meshing application.

   a. Expand the **Sizing** node in the **Details** view to reveal additional sizing parameters.

   b. Select **Off** from the **Use Advanced Size Function** drop-down list.



4. Generate the mesh.

   a. Right-click **Mesh** in the **Outline** tree view, and select **Update** in the context menu.

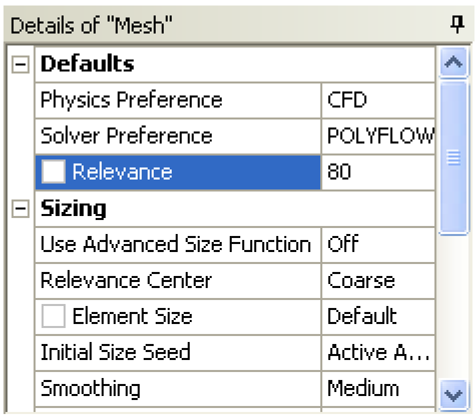
   The geometry window will display the generated mesh.

Note

Using the **Generate Mesh** option from the **Mesh** context menu creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh and creates the relevant mesh files for your project and updates the ANSYS Workbench cell that references this mesh.

b. Refine the mesh.

   i. Enter `80` for **Relevance** under **Defaults** in the **Details** view.



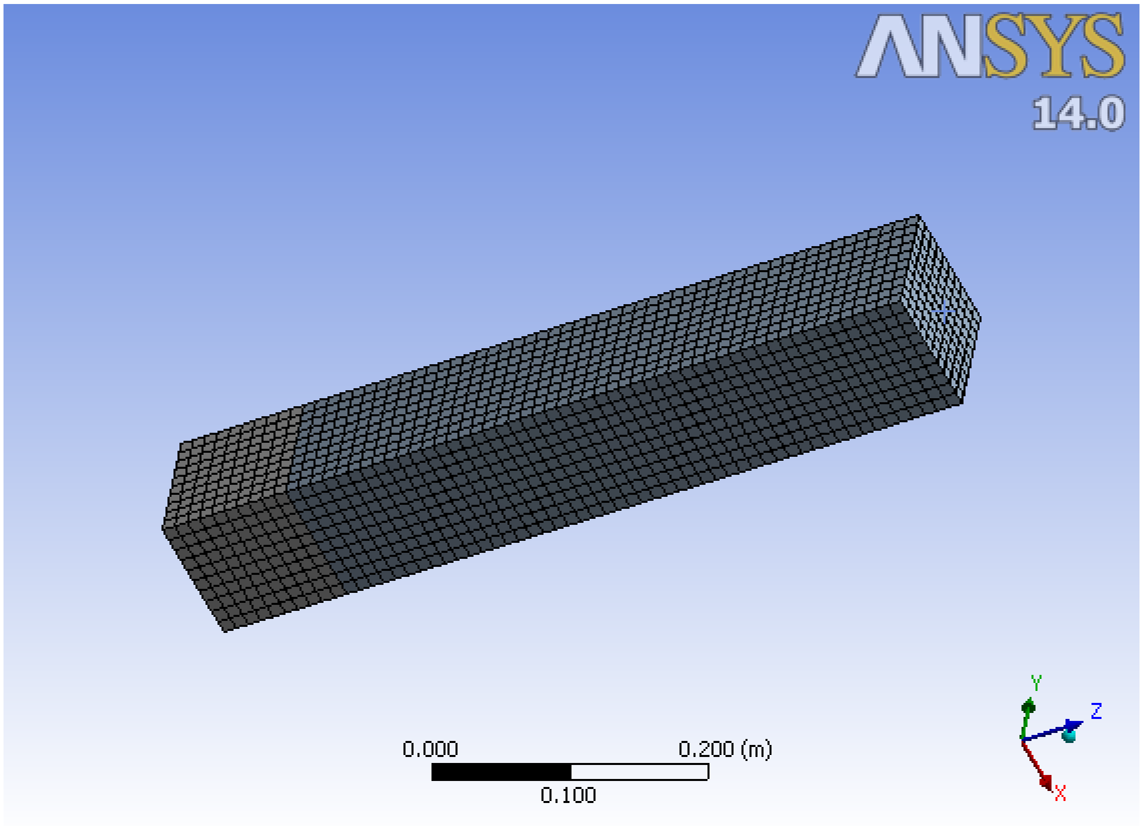
   ii. Right-click **Mesh** in the **Outline** tree view, and select **Update** in the context menu.

The geometry window will display the refined mesh.

Extra

After the mesh is generated, you can view the mesh statistics by expanding the **Statistics** node in the **Details** view to reveal information about the number of nodes, the number of elements, and other details.

Figure 1.16 The Computational Mesh for the Extrusion Geometry



5. Close the ANSYS Meshing application.

   When you close the ANSYS Meshing application, ANSYS Workbench automatically saves the mesh and updates the **Project Schematic** accordingly (the state of the **Mesh** cell changes from **Refresh Required** to **Up-to-Date**, indicating that there is a mesh now associated with the fluid flow analysis system).

6. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

Files

	A
1	Name
4	PFL.agdb
5	PFL.1.poly
6	PFL.mshdb
7	designPoint.wbdp
8	PFL.poly.log
9	PFL.1.poly.log

Note the addition of the mesh files (`PFL.1.poly` and `PFL.mshdb`) to the list of files. The `PFL.1.poly` file was created when you updated the mesh, and the `PFL.mshdb` file was generated when you closed the ANSYS Meshing application.

1.4.5. Step 4: Setting Up the CFD Simulation in ANSYS POLYDATA

In this step, you will proceed to set up a CFD analysis using ANSYS POLYDATA, then review the list of files generated by ANSYS Workbench.

1. Import the data file (`polyflow.dat`).

   The data file you will import has already been set up for a 3D extrusion simulation with a single inlet flow rate. For details on how to set up a similar data file in ANSYS POLYDATA, see Tutorial 6 in the separate Tutorial Guide.

   a. Right-click the **Setup** cell in the `ext3d` fluid flow analysis system, and click **Import POLYFLOW Dat ...** in the context menu that opens.

   b. Use the **Open** dialog box to browse to the `ext3d-workbench` folder you unzipped in a previous step, select `polyflow.dat`, and click **Open**.

      The state of the ***Setup*** *cell remains* ***Refresh Required****, indicating that even though there is a data file now associated with the fluid flow analysis system, you still must perform an update for the cell.*

   c. Right-click on the **Setup** cell and click **Update** in the context menu that opens.

      After ANSYS POLYDATA converts the `.poly` mesh file into a `.msh` file and checks for coherence between the mesh and data files, the state for the ***Setup*** *cell becomes* ***Up-to-Date****. At this point it would be possible to run the ANSYS POLYFLOW solver for your simulation; however, for this tutorial you will first modify the data file.*

2. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

   Files

	A
1	Name
6	PFL.mshdb
7	config_polydata.p3rc
8	polyflow.dat
9	convert.msh
10	designPoint.wbdp
11	polyflow.bat

   Note the addition of the mesh file (`convert.msh`) and the data file (`polyflow.dat`) to the list of files.

3. Start ANSYS POLYDATA.

Double-click the **Setup** cell in the `ext3d` fluid flow analysis system.

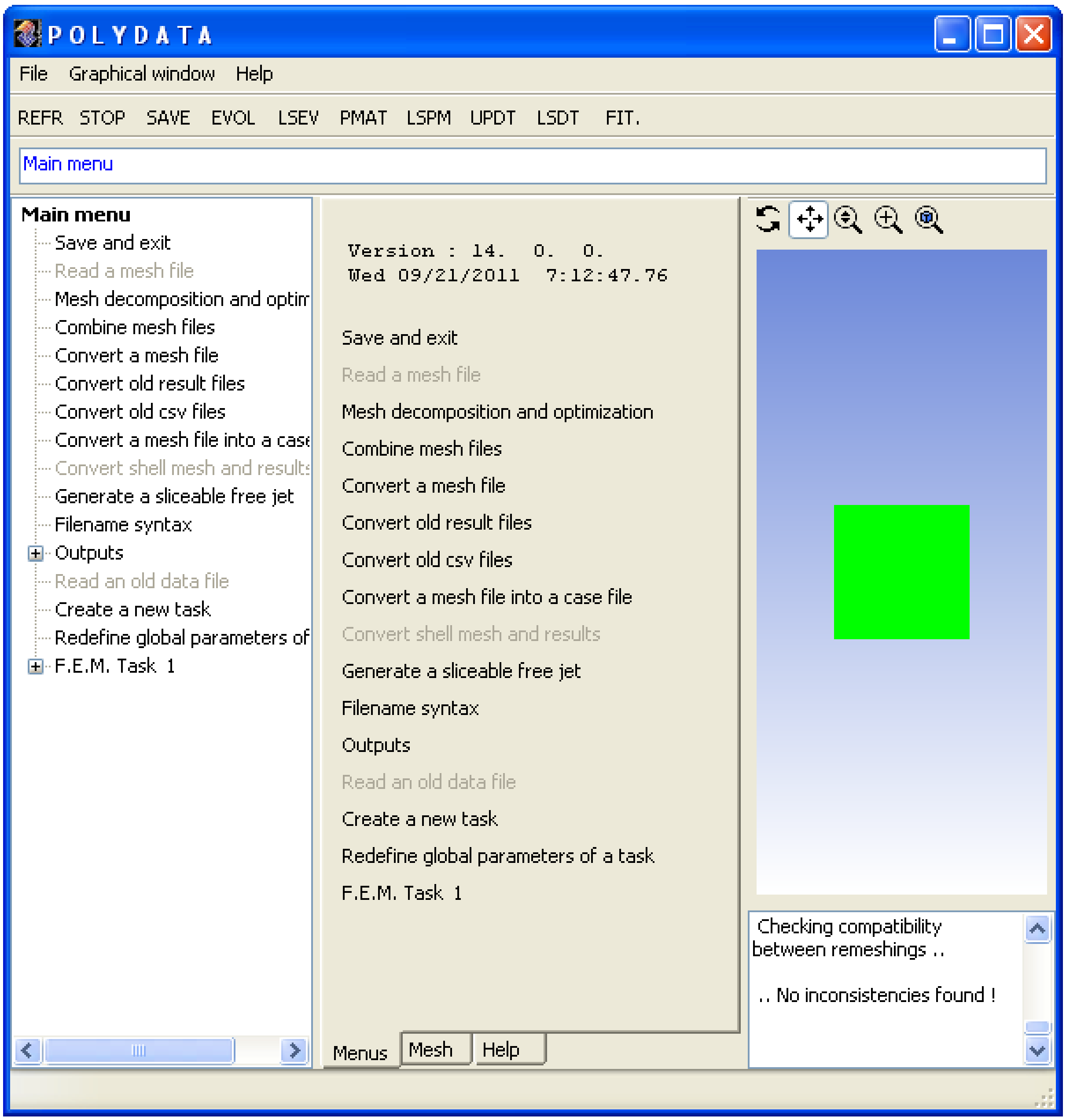
Extra

You can also launch ANSYS POLYDATA by right-clicking on the **Setup** cell and clicking **Edit...** in the context menu that opens.

Note

The mesh is automatically loaded and displayed in the **Graphics Display** window by default.

Figure 1.17 The ANSYS POLYDATA Application



4. Modify the data file so that the inlet flow rate is flagged as modifiable in a user-defined template (UDT).

Note

UDTs are considered input parameters by ANSYS Workbench.

   a. Click **F.E.M. Task 1** in the main **POLYDATA** menu.

      F.E.M. Task 1

   b. Click **3D die swell** (which is the name that was given to the sub-task for the flow problem when the data file was created) in the **F.E.M. Task 1** menu.

      3D die swell

   c. Click **Flow boundary conditions** in the **3D die swell** menu.

      Flow boundary conditions

   d. Select **Inflow along INLET** in the **Flow boundary conditions** menu and click **Modify**.

   e. Click the **UPDT** button at the top of the ANSYS POLYDATA application window, to enable template inputs.

   f. Click **Inflow** in the **Flow boundary condition along INLET** menu.

      Inflow

   g. Retain the selections of **Automatic** and **Volumetric flow rate** in the **Inflow calculation on INLET** menu, and note that the flow rate is already set to 10. Then click **Upper level menu**.

   h. Click **Create a new template entry** in the **Create template entry** menu.

      Create a new template entry

   i. Click the **UPDT** button again at the top of the ANSYS POLYDATA application window, to disable template inputs.

   j. Click **Upper level menu** repeatedly to return to the main **POLYDATA** menu.

5. Save the data file and close ANSYS POLYDATA.

   a. Click **Save and exit** in the main **POLYDATA** menu.

      Save and exit

   b. Click **Accept** in the **Field Management** menu.

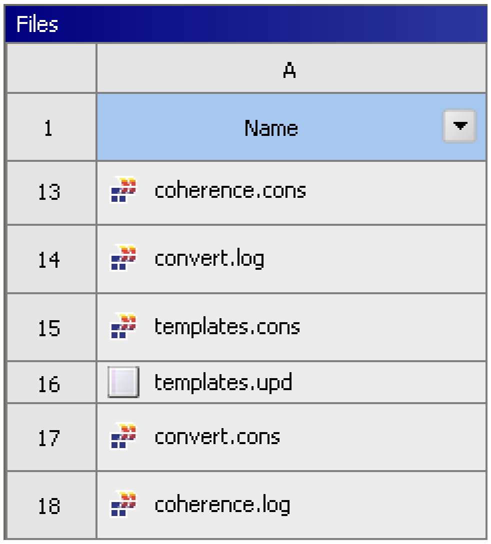
      Accept

   c. Click **Continue**.

      Continue

The **Parameters** *cell will be added to the* `ext3d` *fluid flow analysis system in the ANSYS Workbench* **Project Schematic** *(cell A7). Also, a* **Parameter Set** *bar will be added below the system with an inbound arrow, indicating that an input parameter has been created.*

6. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.



Files

	A
1	Name
13	coherence.cons
14	convert.log
15	templates.cons
16	templates.upd
17	convert.cons
18	coherence.log

Note the addition of the template file (`templates.upd`*) to the list of files.*

1.4.6. Step 5: Running the CFD Simulation in ANSYS POLYFLOW

Now that your mesh and data files are properly set up, in this step you will use the ANSYS POLYFLOW solver to run the initial simulation, generate results files, then review the list of files generated by ANSYS Workbench.

1. Start ANSYS POLYFLOW.

   In the ANSYS Workbench **Project Schematic**, right-click the **Solution** cell in the `ext3d` fluid flow analysis system (cell A5), and click **Update** in the context menu that opens.

   The ANSYS POLYFLOW solver will begin running. When the calculation is complete, the state for the ***Solution*** *cell becomes* ***Up-to-Date***.

2. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

Files	
	A
1	Name
11	cfx.res
12	polyflow.lst
13	res.msh
14	res
15	last_result.pub
16	xml.xml

Note the addition of the listing file (`polyflow.lst`), the ANSYS POLYFLOW results file (`res`), the output mesh file (`res.msh`), and the CFD-Post file (`cfx.res`) to the list of files. For more information about ANSYS POLYFLOW (and the files associated with it), see the ANSYS POLYFLOW 14.0 User's Guide.

1.4.7. Step 6: Displaying Results in ANSYS CFD-Post

In this step, you will use ANSYS CFD-Post to view the results of your initial simulation, create an expression that can be used as an output parameter for ANSYS Workbench, then review the list of files generated by ANSYS Workbench.

1. Start ANSYS CFD-Post.

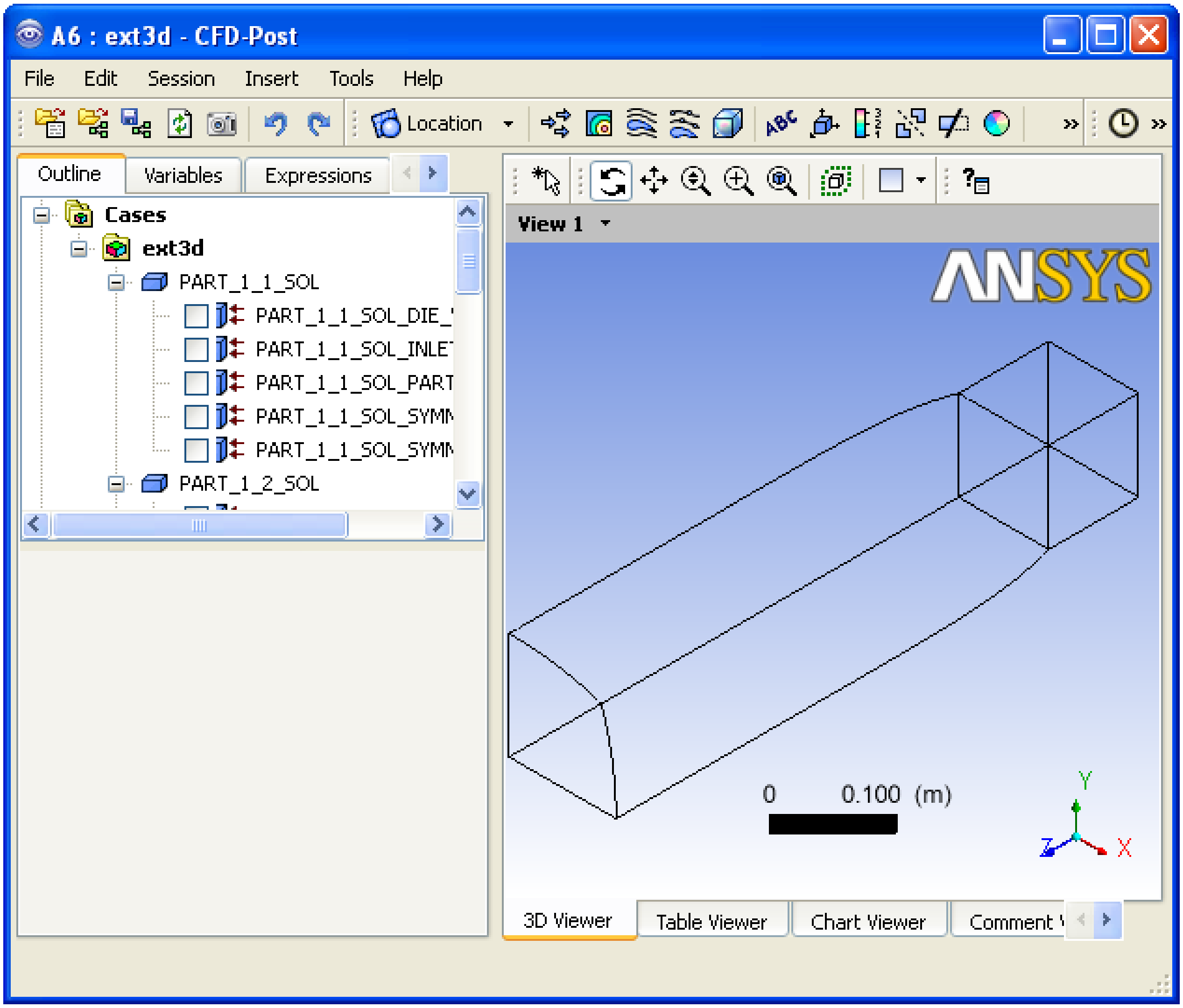
   In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the `ext3d` fluid flow analysis system (cell A6).

   Extra

   You can also start ANSYS CFD-Post by right-clicking the **Results** cell and selecting the **Edit...** option in the context menu that opens.

   The ANSYS CFD-Post application will launch with the extrusion geometry already loaded (displayed in outline mode). Note that ANSYS POLYFLOW results are also automatically loaded into ANSYS CFD-Post.

Figure 1.18 The Extrusion Geometry Loaded into ANSYS CFD-Post

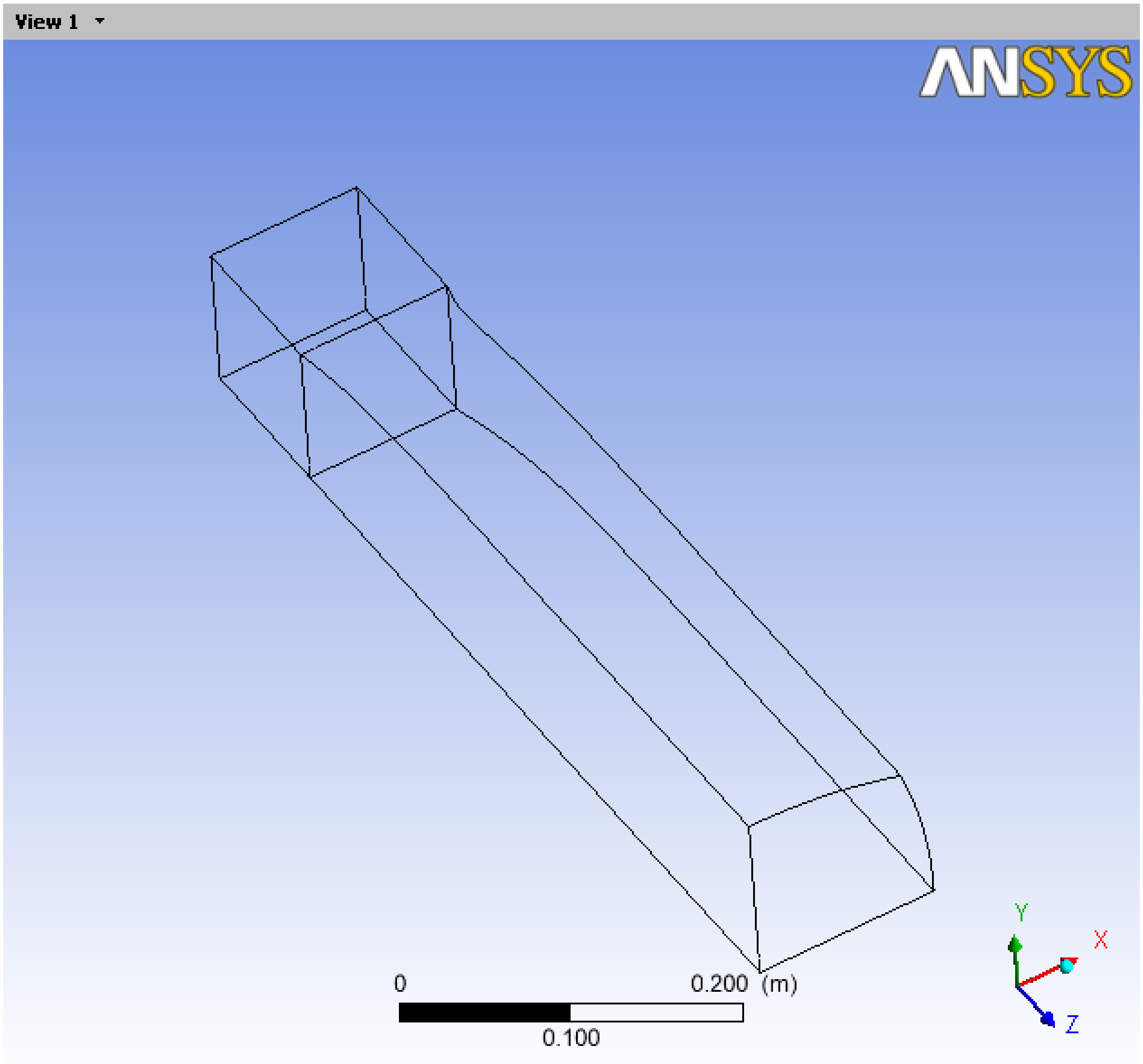


2. Obtain the view shown in *Figure 1.19* (p. 29).

   a. Rotate the view, by holding the center mouse button and moving your pointer in the viewer area.

   b. Reduce the magnification of the view by clicking the **Zoom** icon at the top of the viewer area (), holding the left mouse button, and moving your pointer in the viewer area.

Figure 1.19 Rotating the View



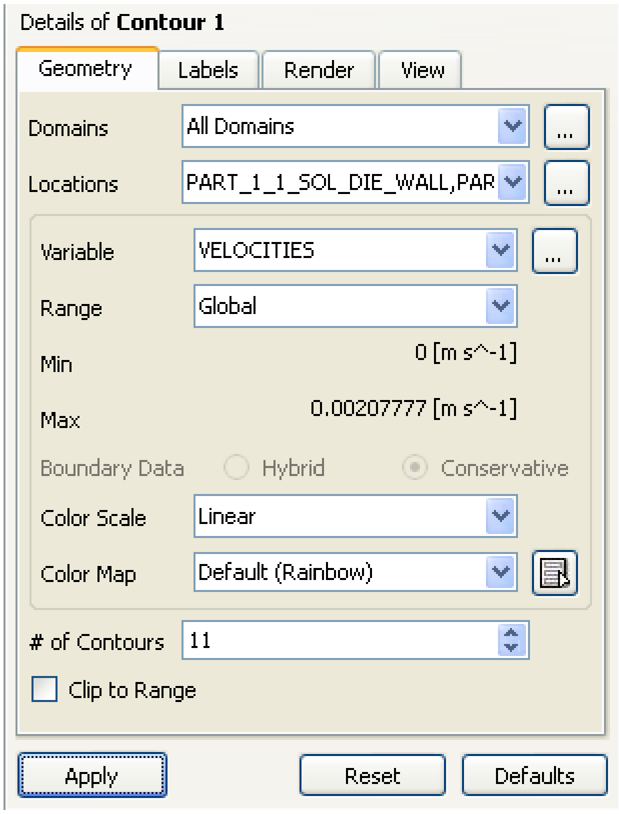
3. Display contours of velocity magnitude on the boundaries (*Figure 1.20* (p. 32)).

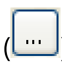
   a. Open the **Insert Contour** dialog box.

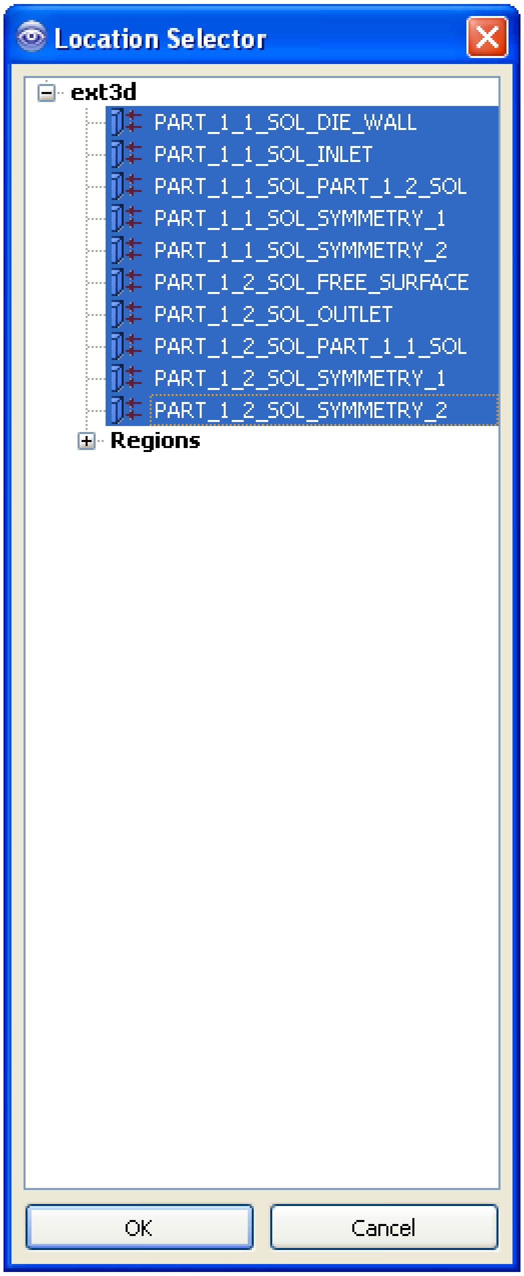
   Insert → Contour

   b. Retain the default entry of **Contour 1** for **Name** and click **OK** to close the dialog box.

   Information about **Contour 1** *will be displayed in the* **Details** *view below the* **Tree** *view in ANSYS CFD-Post. The* **Details** *view contains all of the settings for a contour object.*



c. Open the **Location Selector** dialog box by clicking the location editor button () next to the **Locations** drop-down list in the **Geometry** tab.



i. Select all of the boundaries listed under **ext3d** by clicking the first one in the list (i.e., **PART_1_1_SOL_DIE_WALL**), holding the **Shift** key, and clicking the last one in the list (i.e., **PART_1_2_SOL_SYMMETRY_2**).

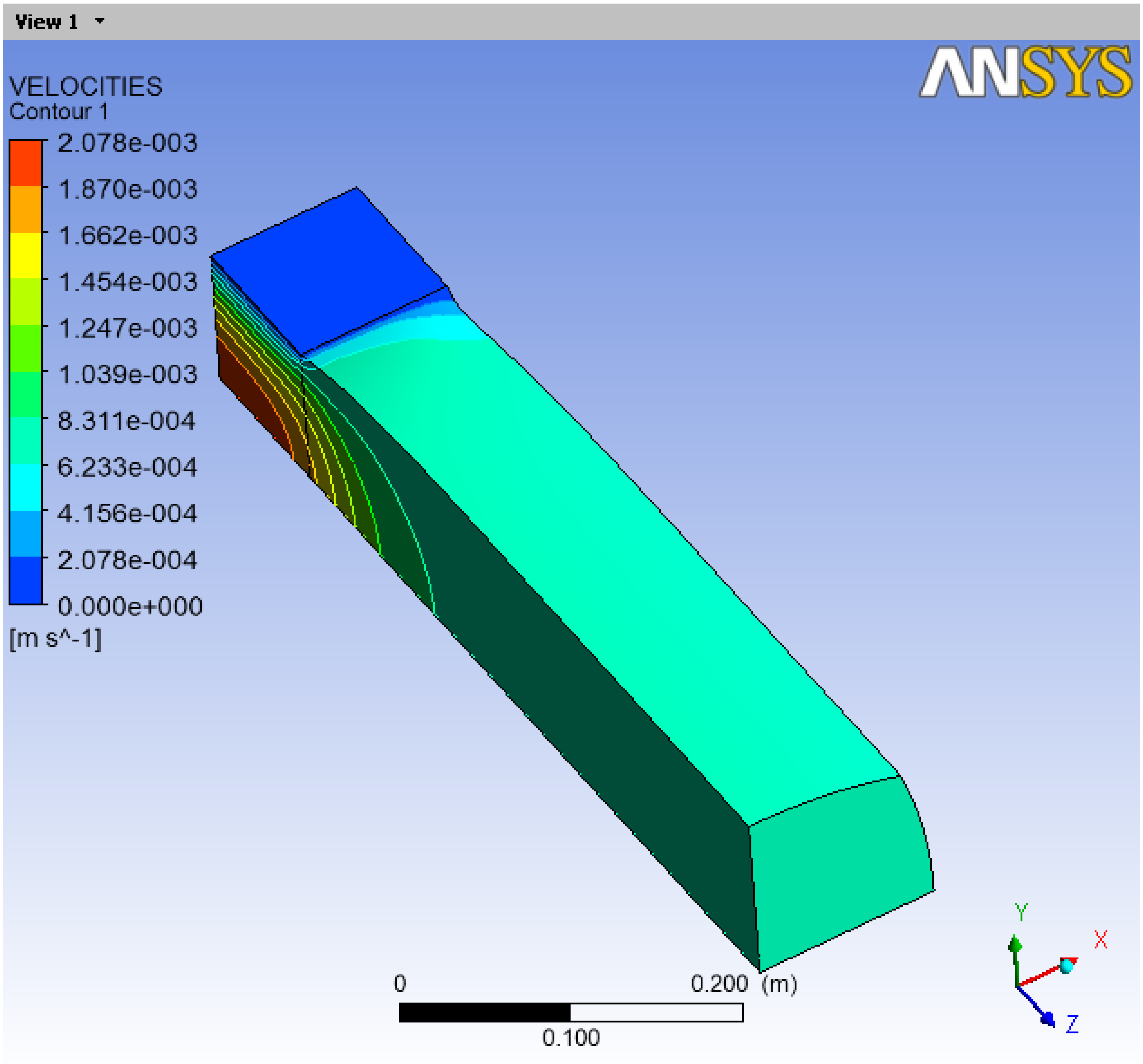
ii. Click **OK** to close the **Location Selector** dialog box.

d. Select **VELOCITIES** from the **Variable** drop-down list.

e. Click **Apply**.

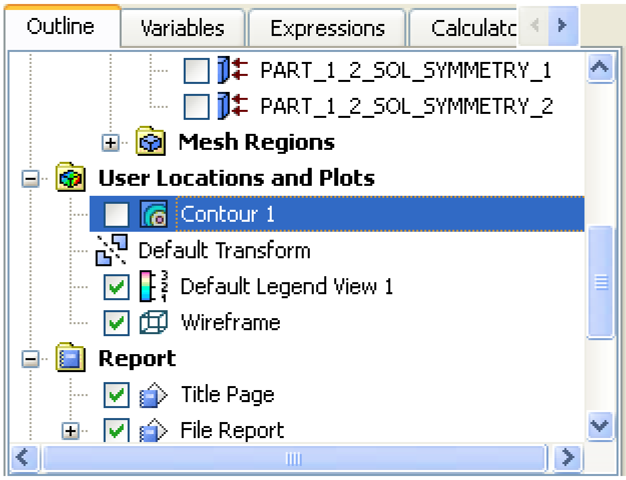
The velocity is 0 along the die wall (as expected) and there is a fully developed profile at the inlet of the die. At the die outlet, the velocity profile changes to become constant throughout the extrudate cross section. The transition between these two states can be seen in the first third of the extrudate.

Figure 1.20 Contours of Velocity Magnitude



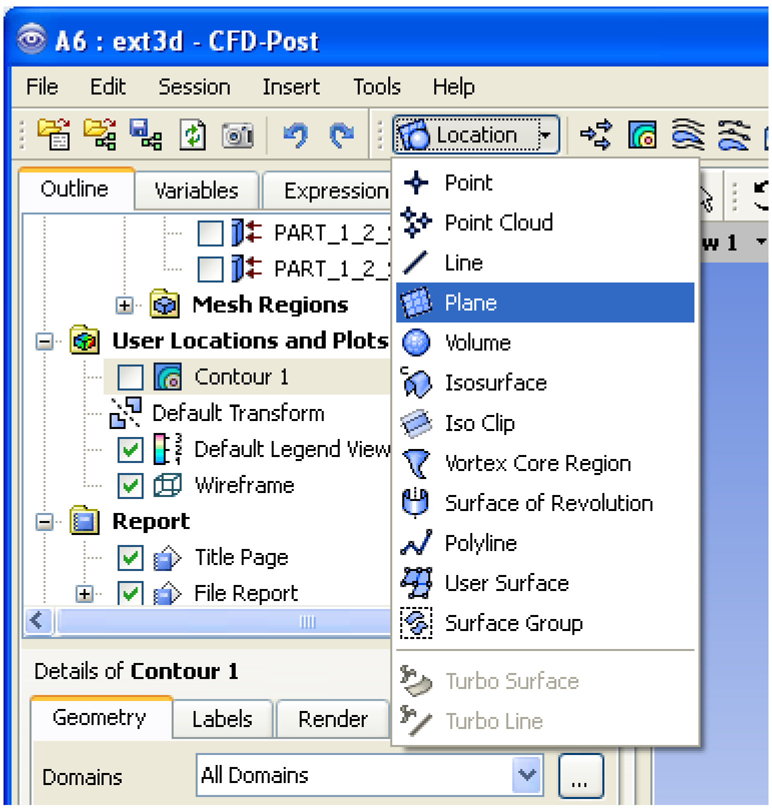
4. Display contours of velocity in cross sections (*Figure 1.21* (p. 37)).

   a. Disable **Contour1** under **User Locations and Plots** in the **Outline** tab of the **Tree** view.



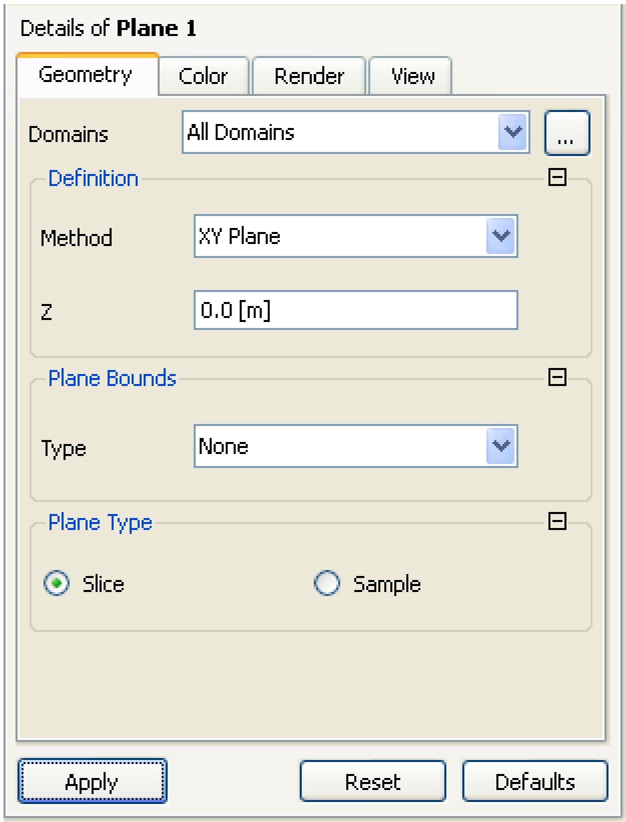
b. Create a cross section plane at $z = 0.0$ m.

i. Select **Plane** from the **Location** drop-down menu, located in the toolbar.



ii. Retain the default entry of **Plane 1** for **Name** in the **Insert Plane** dialog box that opens, and click **OK**.

*Information about **Plane 1** will be displayed in the **Details** view.*



iii. Retain the default selection of **XY Plane** for **Method** in the **Geometry** tab of the **Details** view for **Plane 1**.

iv. Retain the default entry of **0.0** m for **Z**.

v. Click **Apply**

c. In a similar manner, create cross section planes at $z = 0.08$ m, 0.15 m, and 0.45 m named **Plane 2**, **Plane 3**, and **Plane 4** respectively. Note that you will retain the default selection of **XY Plane** for **Method** and enter appropriate values for **Z** in the **Details** view.

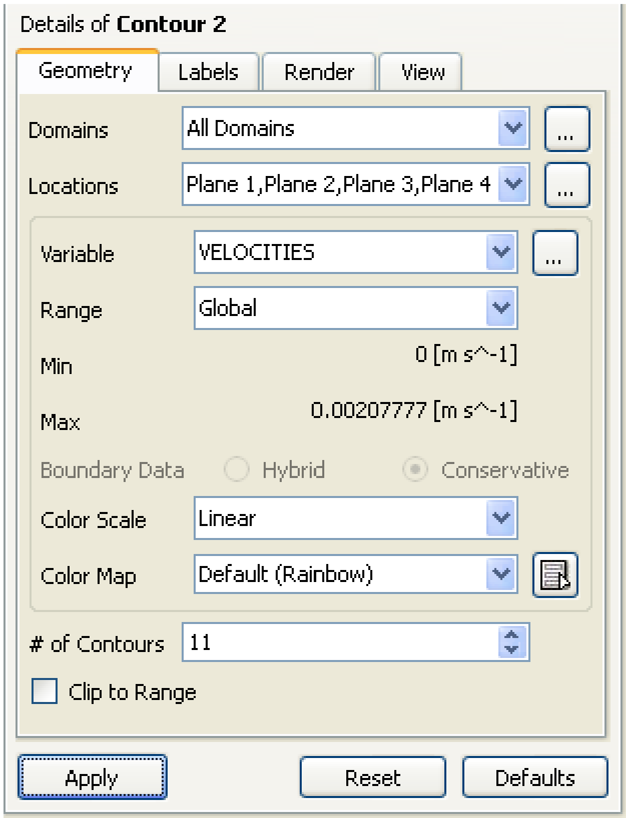
d. Disable **Plane 1**, **Plane 2**, **Plane 3**, and **Plane 4** under **User Locations and Plots** in the **Outline** tab of the **Tree** view, so that the planes are no longer colored gray in the viewer area.

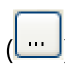
e. Open the **Insert Contour** dialog box.

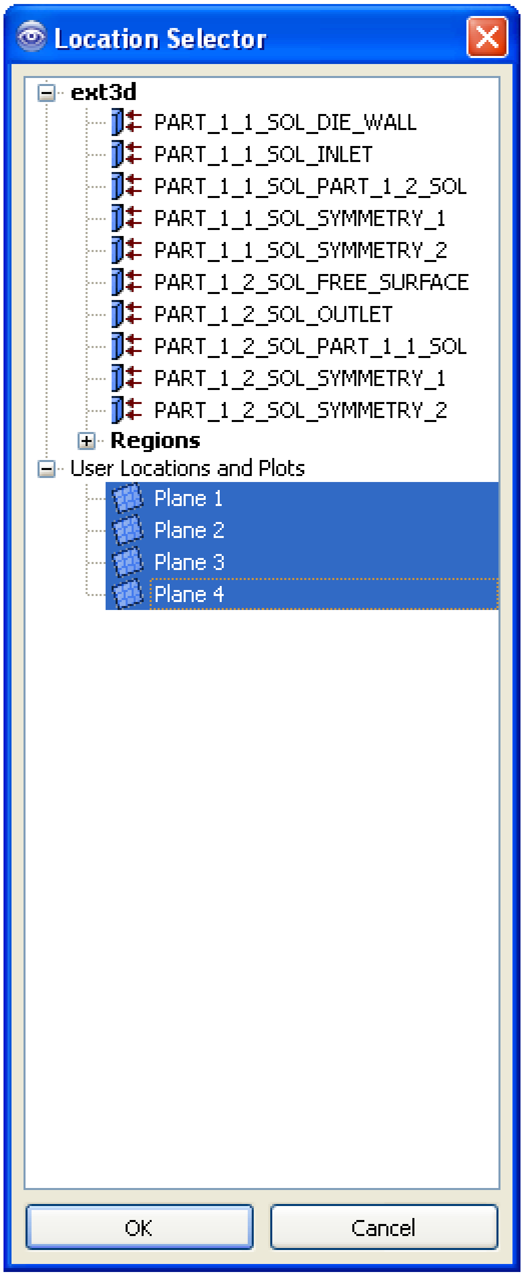
Insert → **Contour**

f. Retain the default entry of **Contour 2** for **Name** and click **OK** to close the dialog box.

Information about **Contour 2** *will be displayed in the* **Details** *view below the* **Tree** *view.*



g. Open the **Location Selector** dialog box by clicking the location editor button () next to the **Locations** drop-down list in the **Geometry** tab.



i. Select all of the planes listed under **User Locations and Plots** by clicking **Plane 1**, holding the **Shift** key, and clicking **Plane 4**.

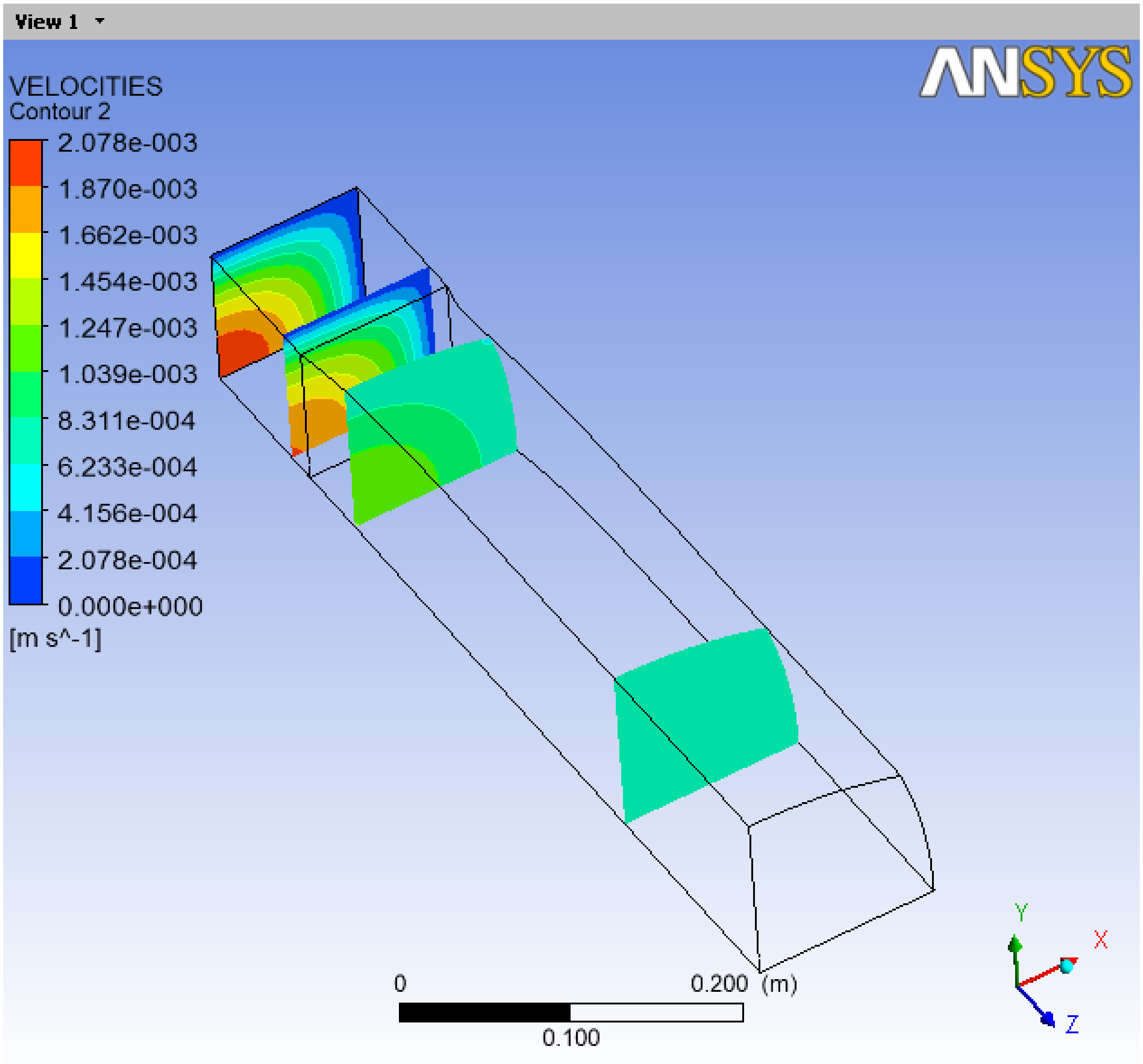
ii. Click **OK** to close the **Location Selector** dialog box.

h. Select **VELOCITIES** from the **Variable** drop-down list.

i. Click **Apply**.

Velocity profiles at the flow inlet, the flow outlet, and planes just before and just after the die exit are displayed. Compare the velocity profile within the die to the velocity profile just after the die exit at the end of the computational domain. In the die the flow is fully developed. The velocity profile is flat (i.e., all the particles in the cross section are at the same velocity) in the extrudate, far away from the die exit. In the transitional zone just beyond the die exit, the velocity profile is reorganized. The velocity profile on the plane $z = 0.15$ *m is no longer fully developed, but it is not yet flat either. The velocity rearrangement is the source of the deformation of the extrudate.*

Figure 1.21 Velocity Profiles at Cross Sections

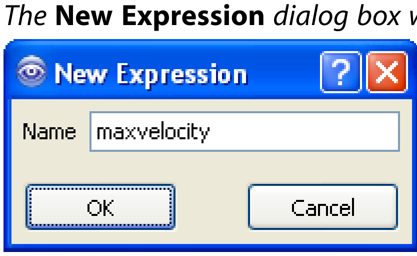


5. Create an expression for the maximum velocity at the flow exit, which can be used as an output parameter in ANSYS Workbench.

   a. Click the **Expressions** tab in the **Tree** view.

   b. Right-click anywhere in the **Expressions** tab and click **New ...** in the menu that opens to create a new expression.

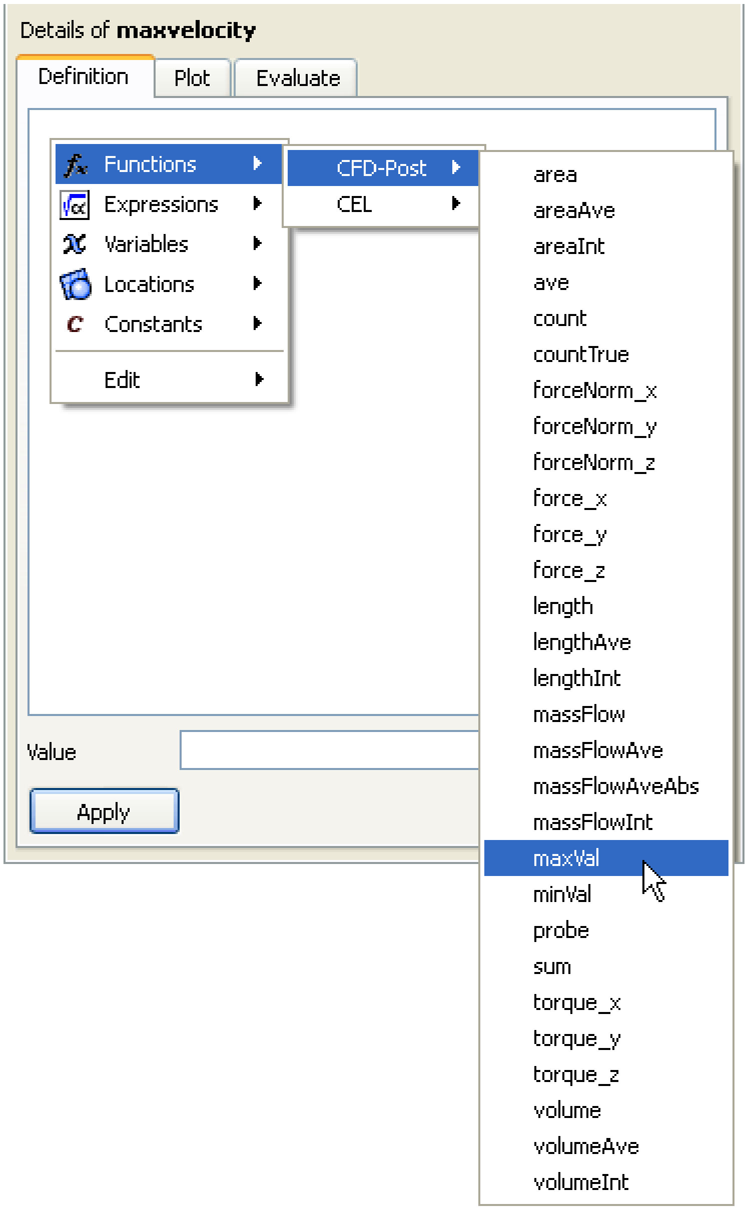
      The **New Expression** *dialog box will open.*

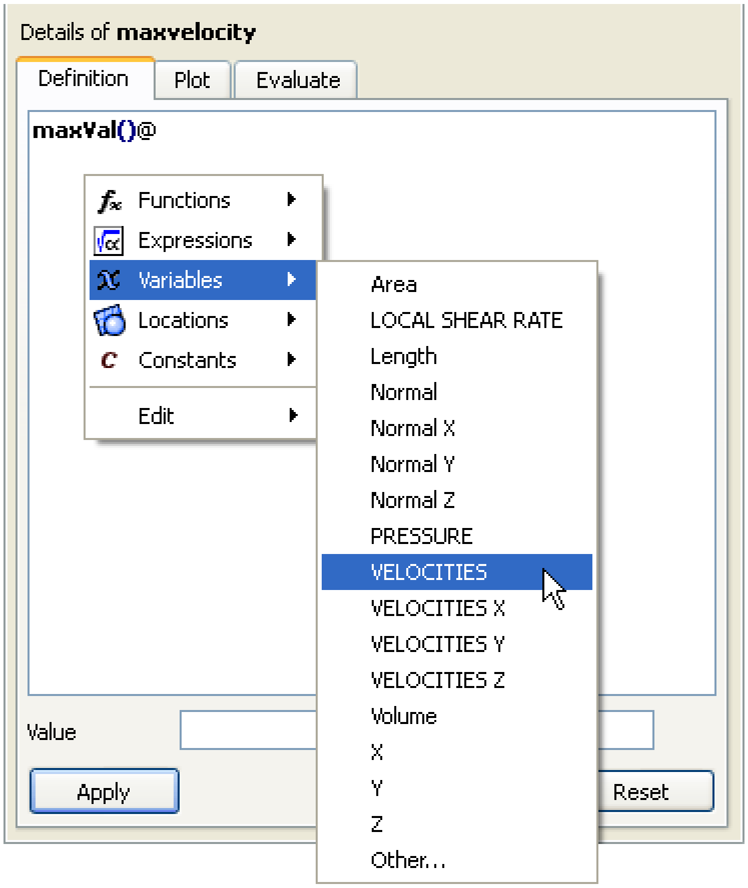
      i. Enter `maxvelocity` for **Name**.

ii. Click **OK** to close the **New Expression** dialog box.

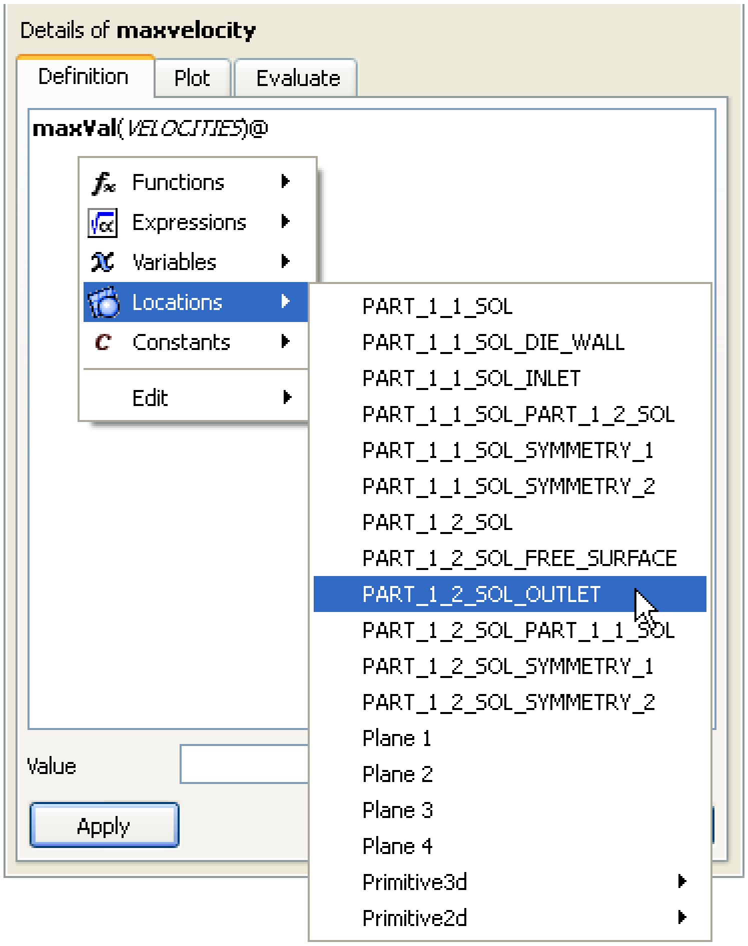
c. Right-click in the **Definition** tab of the **Details** view, move your pointer over **Functions**, move your pointer over **CFD-Post**, and click **maxVal**, to specify that the function in the expression obtains the maximum value.



d. Make sure that the cursor is between the parentheses of **maxVal** ()@, right-click in the **Details** view again, move your pointer over **Variables**, and click **VELOCITIES**, to specify that the variables obtained in the expression are velocities.



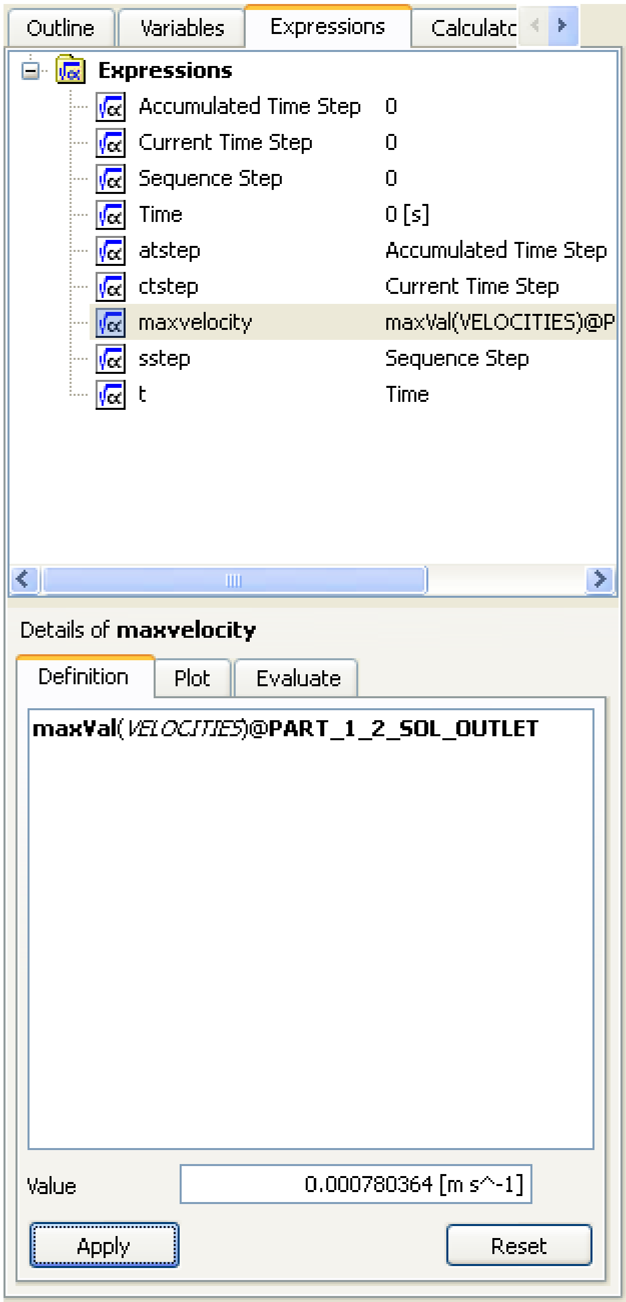
e. Move the cursor so that it is after the @ symbol of **maxVal** (*VELOCITIES*)@, right-click in the **Details** view again, move your pointer over **Locations**, and click **PART_1_2_SOL_OUTLET**, to specify that the variables are obtained for the expression at the flow exit.



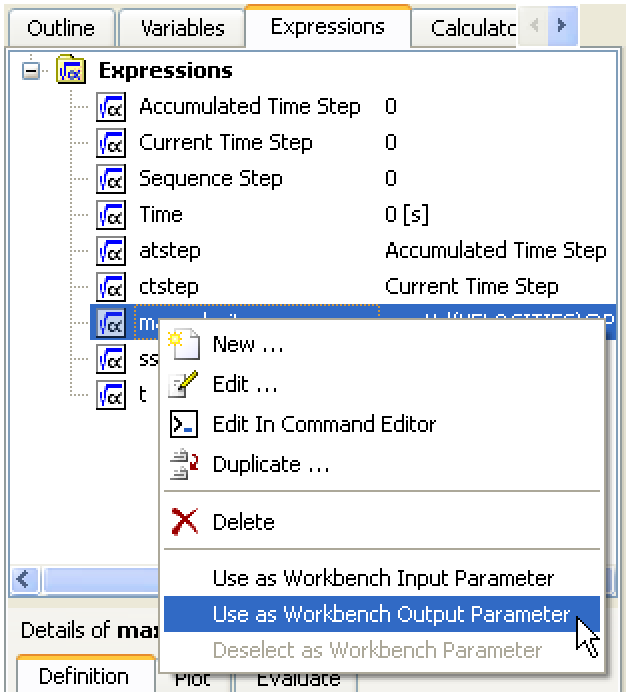
f. Click **Apply**.

The expression in the **Definition** *tab of the* **Details** *view will be defined as* **maxVal** *(VELOCITIES)@* **PART_1_2_SOL_OUTLET** *with a* **Value** *of approximately* `7.8 X 10`$^{-4}$ `m/s`*, and* **maxvelocity** *will be added to the list in the* **Expressions** *tab of the* **Tree** *view, as shown in* *Figure 1.22 (p. 41)*.

Figure 1.22 Creating an Expression for an Output Parameter



g. Right-click on **maxvelocity** in the **Expressions** tab of the **Tree** view and select **Use as Workbench Output Parameter** in the context menu that opens.



An outbound arrow will be added from the **Parameters** *cell to the* **Parameter Set** *bar in the* **Project Schematic**, *indicating that an output parameter has been created.*

6. Close the ANSYS CFD-Post application.

Note

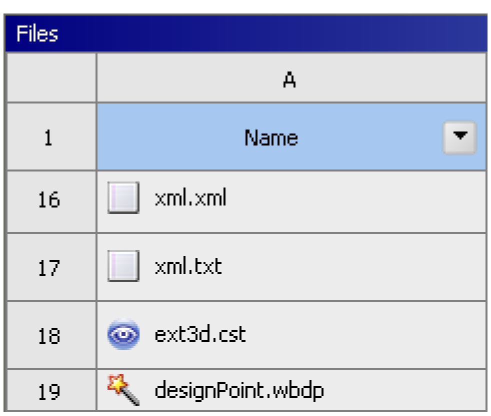
Note that the ANSYS CFD-Post state files are automatically saved when you exit ANSYS CFD-Post and return to ANSYS Workbench.

7. Save the `ext3d-wb` project in ANSYS Workbench.

   File → **Save**

8. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

Figure 1.23 Displaying the Files View after Viewing Results in ANSYS CFD-Post



Note the addition of the ANSYS CFD-Post state file (`ext3d.cst`) to the list of files. For more information about ANSYS CFD-Post (and the files associated with it), see the ANSYS CFD-Post documentation.

1.4.8. Step 7: Exploring Additional Solutions

At this point you have run the simulation with an initial inlet flow rate. In this step, you will create multiple design points for various inlet flow rates, solve them with a single action, then review the list of files generated by ANSYS Workbench.

Note

ANSYS DesignXplorer is licensed separately from ANSYS POLYFLOW. If you do not have access to ANSYS DesignXplorer, you will not be able to perform some of the steps that follow, such as computing multiple design points or plotting results in a chart.

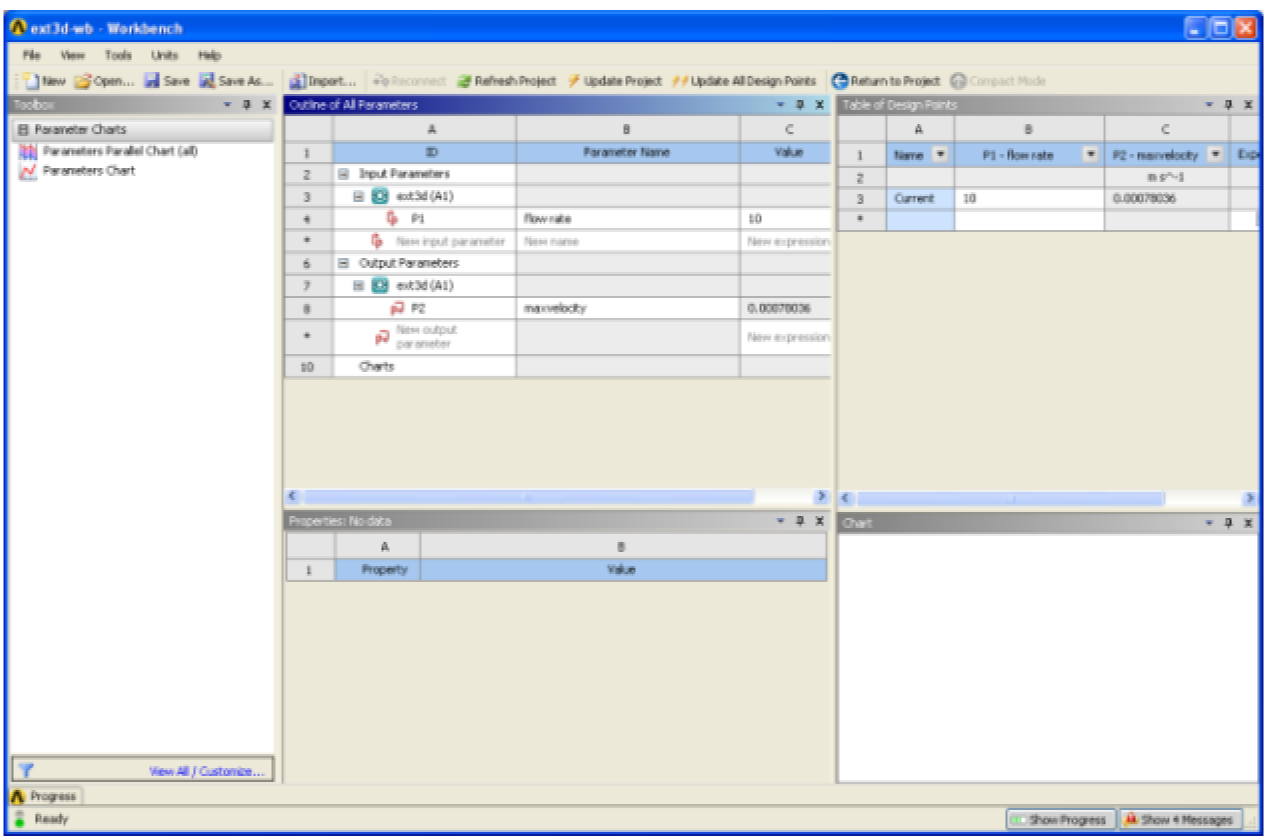
1. Open the **Parameters and Design Points** view (*Figure 1.24* (p. 43)).

   In the ANSYS Workbench **Project Schematic**, double-click the **Parameter Set** bar below the `ext3d` fluid flow analysis system.

   Extra

   You can also open the **Parameters and Design Points** view by right-clicking the **Parameter Set** bar and selecting the **Edit...** option in the context menu that opens.

Figure 1.24 The Parameters and Design Points View



2. Run the calculation again with a new inlet flow rate for the current design point.

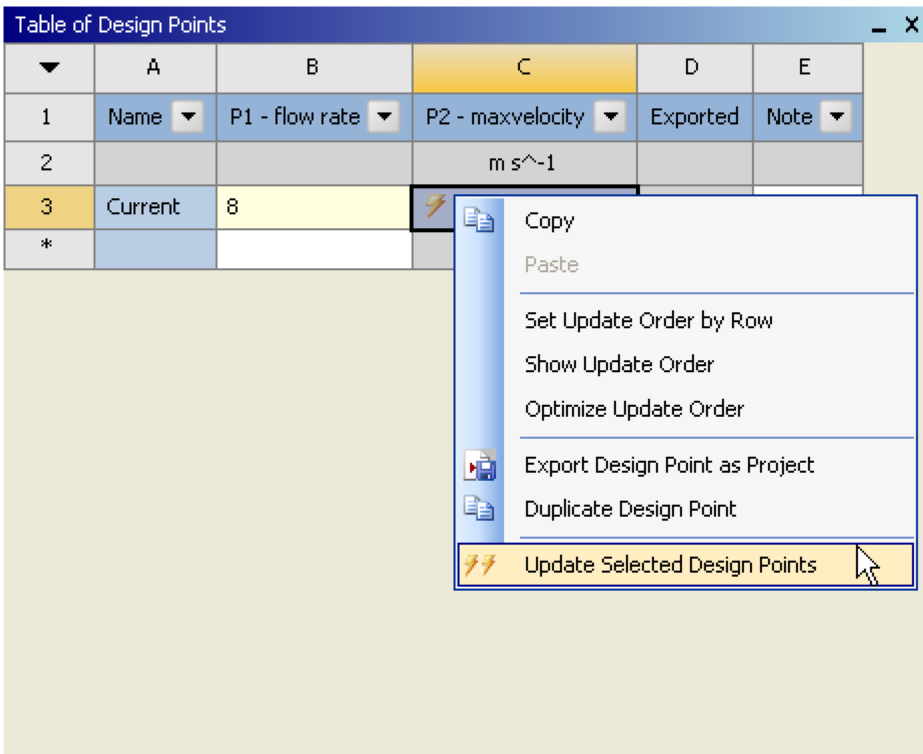
   a. Enter 8 under **P1 - flow rate** for the **Current** design point (i.e., cell **B3**) in the **Table of Design Points**.

      *An **Update Required** icon will be added to the cell under **P2- maxvelocity** for the **Current** design point (i.e., cell **C3**).*

   b. Right-click on the cell under **P2 - maxvelocity** for the **Current** design point and select **Update Selected Design Points** in the context menu that opens, to generate the maximum velocity at the flow exit with the revised inlet flow rate.

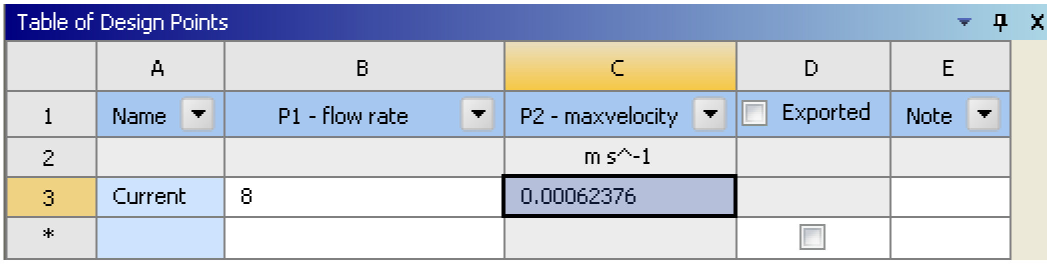
      Extra

      You can also update the design point by clicking **Update Project** in the ANSYS Workbench toolbar.



      *A dialog box will open to inform you that some open editors may close during this process. Click **OK** to proceed.*

*ANSYS POLYDATA will update the data file based on the revised inlet flow rate and ANSYS POLYFLOW will run again. When the calculation is complete, the **Table of Design Points** will display a new value of approximately* 6.2×10^{-4} m/s *under **P2 - maxvelocity** for the **Current** design point.*



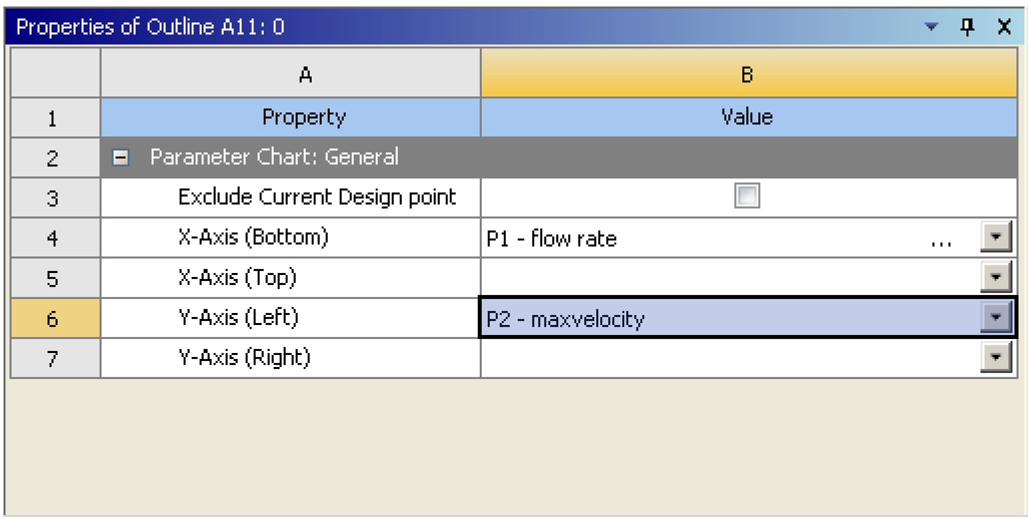
3. Create a chart for the updated current design point.

   a. Click **P1** under **Input Parameters** (i.e., cell **A4**) in the **Outline of All Parameters**.

      The ANSYS Workbench **Toolbox** *will display options for* **Parameter Charts**.

   b. Double-click **Parameters Chart P1 vs ?** in the **Toolbox** to open the **Properties of Outline A11:0** window at the bottom of the **Parameters and Design Points** view.

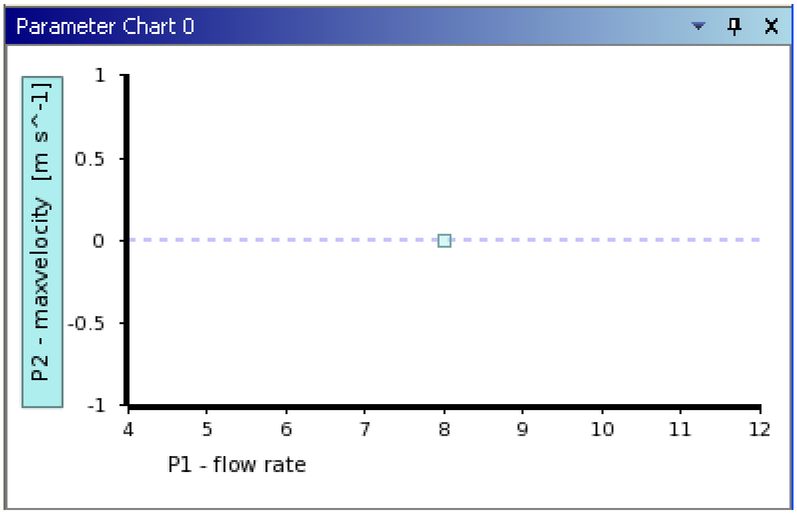
      The **Properties of Outline A11:0** *window will display an initial setup for* **Parameter Chart 0**, *in which* **P1 - flow rate** *is selected from the* **X-Axis (Bottom)** *drop-down list.*



   c. Select **P2-maxvelocity** from the **Y-Axis (Left)** drop-down list in the **Properties of Outline A11:0** window.

The current design point will be plotted in **Parameter Chart 0** *(Figure 1.25 (p. 46)).*

Figure 1.25 The Chart of the Current Design Point



4. Create more design points for a range of inlet flow rates.

   a. Enter `10` for **P1 - flow rate** in the row beneath the **Current** design point (i.e., cell **B***) in the **Table of Design Points**, so that a new row is added (**4**) with **DP 1** as the **Name**.

Table of Design Points

	A	B	C	D	E
1	Name	P1 - flow rate	P2 - maxvelocity	Exported	Note
2			m s^-1		
3	Current	8	0.00062376		
4	DP 1	10			
*					

   b. In a similar manner, create additional design points **DP 2** and **DP 3** with a **P1 - flow rate** of `11` and `12`, respectively.

Extra

You can specify that the data generated for any of the added design points is saved in a separate project file (e.g., `ext3d-wb_dp1.wbpj`) by enabling the **Exported** option in column **D** for the design points.

5. Generate the values for the maximum velocity at the flow exit for all of the new design points.

   Click **Update All Design Points** in the ANSYS Workbench toolbar.

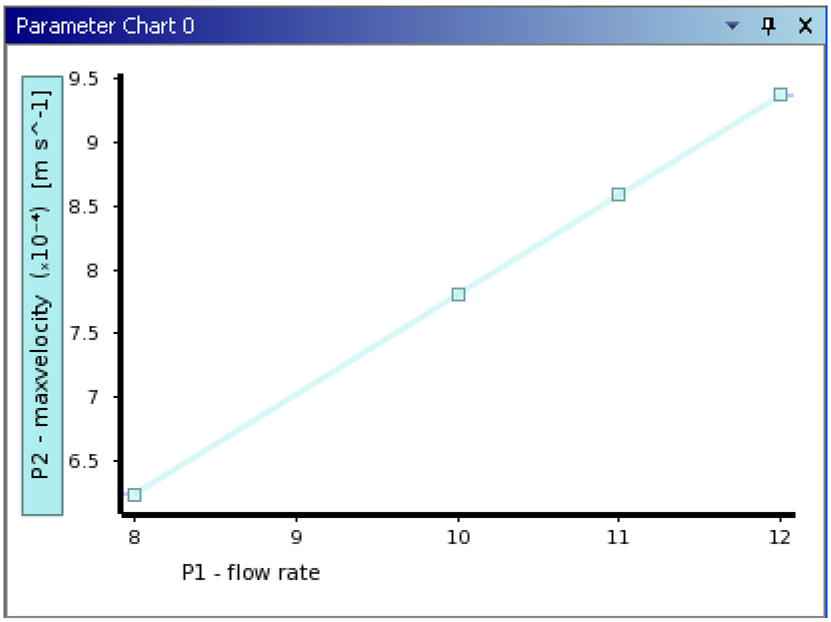
   ANSYS POLYDATA will update and ANSYS POLYFLOW will run repeatedly to solve for each of the design points. As each calculation completes, the **Table of Design Points** *(Figure 1.26 (p. 47)) and* **Parameter Chart 0** *(Figure 1.27 (p. 47)) will be updated.*

Figure 1.26 Displaying Values for All of the Design Points

Table of Design Points

	A	B	C	D	E
1	Name	P1 - flow rate ...	P2 - maxvelocity	Exported	Note
2			m s^-1		
3	Current	8	0.00062376		
4	DP 1	10	0.00078036	☐	
5	DP 2	11	0.00085872	☐	
6	DP 3	12	0.00093711	☐	
*				☐	

Figure 1.27 The Chart of All of the Design Points



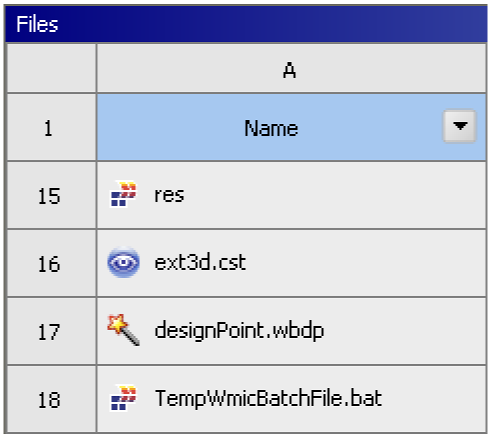
6. Save the `ext3d-wb` project in ANSYS Workbench.

   File → **Save**

7. Return to the **Project Schematic** view by clicking the **Return to Project** button in the ANSYS Workbench toolbar.

8. View the files generated by ANSYS Workbench, as displayed in the **Project Schematic**.

Figure 1.28 Displaying the Files View after Exploring Solutions



Note that the list of files shows that the design point file (`designPoint.wbdp`*) was updated. For more information about the files associated with ANSYS Workbench, see the ANSYS Workbench documentation.*

1.4.9. Step 8: Summary

In this tutorial, portions of ANSYS Workbench were used to simulate a 3D extrusion and to compare the flow exit velocities associated with a range of inlet flow rates.

ANSYS DesignModeler was used to prepare the geometry, ANSYS Meshing was used to create a computational mesh, ANSYS POLYDATA was used to set up the simulation, ANSYS POLYFLOW was used to calculate the fluid flow throughout the geometry using the computational mesh, and CFD-Post was used to analyze the results. In addition, the **Parameters and Design Points** view of ANSYS Workbench was used to add additional design points and compare their associated flow exit velocities on a chart.