

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



Copyright and Trademark Information

© 2011 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFID-ENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

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

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. ANSYS POLYFLOW in ANSYS Workbench Tutorial: 3D Extrusion	
1.1. Introduction	1
1.2. Prerequisites	
1.3. Problem Description	
1.4. Setup and Solution	3
1.4.1. Preparation	3
1.4.2. Step 1: Creating a Fluid Flow Analysis System in ANSYS Workbench	
1.4.3. Step 2: Preparing the Geometry in ANSYS DesignModeler	8
1.4.4. Step 3: Meshing the Geometry in the ANSYS Meshing Application	11
1.4.5. Step 4: Setting Up the CFD Simulation in ANSYS POLYDATA	23
1.4.6. Step 5: Running the CFD Simulation in ANSYS POLYFLOW	
1.4.7. Step 6: Displaying Results in ANSYS CFD-Post	27
1.4.8. Step 7: Exploring Additional Solutions	
1.4.9. Step 8: Summary	48

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.

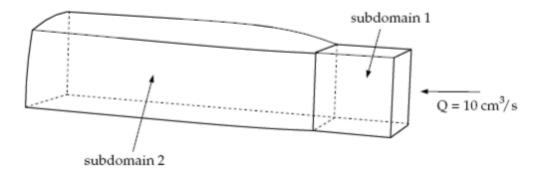
1.3. Problem Description

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 $Q = 10 \text{ cm}^3/\text{s}$ (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

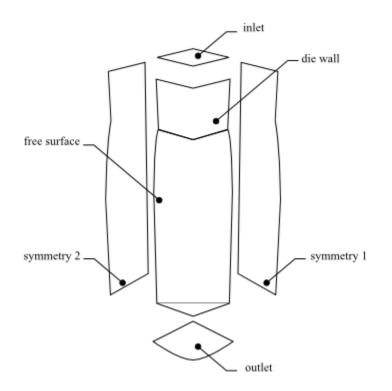


The incompressibility and momentum equations are solved over the computational domain. The domain for the problem is divided into two subdomains (as shown in *Figure 1.1* (p. 2)) so that a remeshing algorithm can be applied only to the portion of the mesh that will be deformed. Subdomain 1 represents 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 $Q = 10 \text{ cm}^3/\text{s}$
- die wall: zero velocity
- free surface: free surface
- symmetry 1: symmetry plane
- symmetry 2: symmetry plane
- outlet: flow exit





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 ext3dworkbench 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**.

🔥 Unsaved Project - Workbench
File View Tools Units Help
👔 New 📴 Open 🛃 Save 🔣 Save As 👔 Import 🍣 Reconnect 🖡
Toolbox • • • X ProjectSchematic
Analysis Systems
S Design Assessment
() Electric
🕟 Explicit Dynamics
💽 Fluid Flow - Blow Molding (POLYFLOW)
🔇 Fluid Flow - Extrusion (POLYFLOW)
🕄 Fluid Flow (CFX)
C Fluid Flow (FLUENT)
G Fluid Flow (POLYFLOW)
Narmonic Response
With a second se
Hydrodynamic Time Response
Linear Buckling
0 Magnetostatic Modal
Modal Modal (Samcef)
Random Vibration
Response Spectrum
Rigid Dynamics
Shape Optimization
w Static Structural
🐷 Static Structural (Samcef)
🛐 Steady-State Thermal
🕐 Thermal-Electric
🚾 Transient Structural
🔃 Transient Thermal
Component Systems
O AUTODYN
🚰 BladeGen
🕘 CFX
Segmeering Data
Explicit Dynamics (LS-DYNA Export)
External Connection
View All / Customize
Ready

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

Oursaved Project - Workbench		
File View Tools Units Help		
Dies Officer I fam II fam be Street	et 🖓 Reconnect 😹 Refresh Project 🍼 Update Project 🔞 Project 🚱 Compact Mode	
	h cjectSchematic	~ ? X
Analysis Systems		
1 Design Assessment		
😝 Electric	- A	
Explicit Dynamics	1 3 Flad Flow (POLYFLOW)	
Fluid Flow - Blow Molding (POLYFLOW)	2 🎯 Geometry 📪	
Fluid Flow - Extrusion (POLYFLOW)	3 🍘 Mesh 😨	
Fluid Flow (OPI)	4 🏨 Sehap 🛛 😨 🖌	
Plud Flow (PLUENT)	S 🔬 Solution 💡	
Plud Flow (POLYFLOW)		
Harmonic Response Hydrodynamic Diffraction	6 🥩 Results 🛛 🖓 🖌	
Hydrodynamic Diffraction Hydrodynamic Time Response	Fluid Flow (POLYFLOW)	
Cinear Bucking		
Magnetostatic		
Modal		
Modal (Sancef)		
Random Vibration		
Response Spectrum		
Rigid Dynamics		
Shape Optimization		
Satic Structural		
Static Structural (Sancel)		
1 Steady-State Thermal		
1 Themal-Electric		
Transient Structural		
Ctansiert Thermal		
Component Systems		
S AUTODIN		
🛃 EladeGen		
e) on:		
Engineering Data		
Explicit Dynamics (LS-D1NA Export)		
Vew Al / Custonize Messages		
Ready	Show Progress 🔅 Show	0 Messages

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 (²) 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** (12) 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 (³) 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.

$\textbf{File} \rightarrow \textbf{Save}$

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.

$View \rightarrow Files$

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

A ext3d-wb - Workbench							5	
File View Tools Units Help								
				0	A			
New 🔐 Open 🛃 Save 🔣 Save As			Update Project	Gueres	Compact Mode			
Toobox • 9 x	Project5	chenatic						* 3 X
El Analysis Systems								
👩 Design Assessment								
😥 Electric		• A	1					
Explicit Dynamics		1 S Fluid Flow (POLYFLOW)						
Fluid Flow - Blow Molding (POLYFLOW)		2 🥔 Geometry 📪						
Fluid Flow - Extrusion (POLYFLOW)		3 🥔 Mesh 💡	1					
S Fluid Flow (OP:0)								
Flud Flow (FLUENT)								
Flud Flow (POLYFLOW)		5 🗌 Solution 🛛 🦓						
Marmonic Response		6 🥩 Results 🛛 💡 🖌	d in the second s					
Hydrodynamic Diffraction		ext3d						
R Hydrodynamic Time Response								
D Linear Buckling								
Magnetostatic								
Modal								
Modal (Sancef) Random Vibration								
Response Spectrum								
Rigid Dynamics								
Shape Optimization								
Satic Structural	Files							* 7 X
Static Structural (Sancel)	and an other states	A		c	p	ε	F	
1 Steady-State Thermal				-	-			
871 Thermal-Electric	1	Name 💌	Cel 10 💌	Sae 💌	Туре 💌	Date Modified	 Location 	•
Transient Structural	2	🔥 ext3d-wb.wbpi		122 KB	ANSYS Project File	2/23/2011 9:23:38 AM	C/(Tutorials	
E Transiert Thermal	3	🍕 designPoint.wbdp		21 KB	Design Point File	2/23/2011 9:23:39 AM	C:(Tutorials)ext3d-wb_files)dp0	
El Component Systems								
C AUTODIN								
🚑 BladeGen								
(II) O'X								
Engineering Data								
Explicit Dynamics (LS-01NA Export)								
View All / Custonize								
A Messages								
Ready							Show Progress Show 0 Me	ssages .

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.

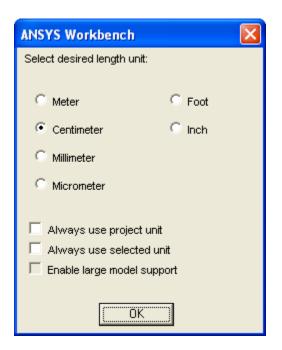
Double-click the **Geometry** cell in the ext3d fluid flow analysis system to launch the ANSYS DesignModeler application.

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.

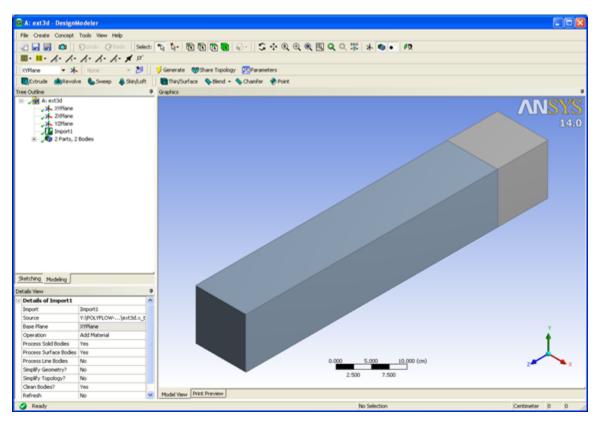
3. Set the units in ANSYS DesignModeler.

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.

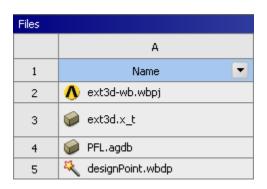
- a. Expand the **2 Parts, 2 Bodies** node.
- b. Click **1** so that it is highlighted.
- c. Hold the **Ctrl** key and click **2** so that it is highlighted as well.
- d. Right-click the highlighted objects and click Form New Part in the menu that opens.

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**.



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

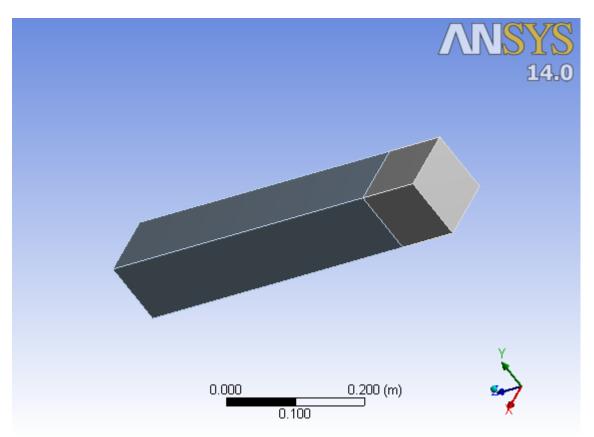
You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.



🖬 A : ext3d - Meshing [/	ANSYS ICEM CFD]					
File Edit View Units Too	File Edit View Units Tools Help 📋 📁 Generate Mesh 🏥 📷 🖪 🐼 🖝 🌒 Worksheet il					
] 📽 👯 💽 - 🖏 - 🕅	🖪 🖪 🚳 - 💪 💠 🤍 🗨 🔍 🖳 🔍 🖧 🗐 🗖 🗖					
」 戸 Show Vertices 🖧 Wirel	rame 🛛 💶 Edge Coloring 👻 🄏 🗸 🏒 🕹 🎝 👻 🎝 👻 🕺 🖓 🖬 🖓 🖬 Hicken Annotat					
Model 🛛 👔 Virtual Topology	Symmetry 🕸 Connections 🚳 Mesh Numbering 🍄 Named Selection					
Outline Project Model (A3) Model (A3) Outline Outline <t< th=""><th>I4.0</th></t<>	I4.0					
< >						
Details of "Model" 4						
Lighting Ambient 0.1						
Diffuse 0.6						
Specular 1	Y					
Color	0.000 0.200 (m) _ 🙏 💡					
	0.100 C.200 (m) ZXX 0.100 C.200 (m) ZXX					
Press F1 for Help	💭 No Messa No Selection Metric (m, kg, N, s, V, A) C					

- 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.

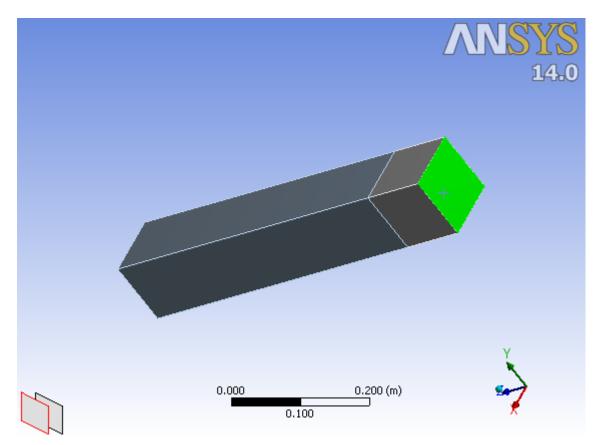
Information will be displayed about the mesh in the **Details** view below the **Outline** tree view.

Note

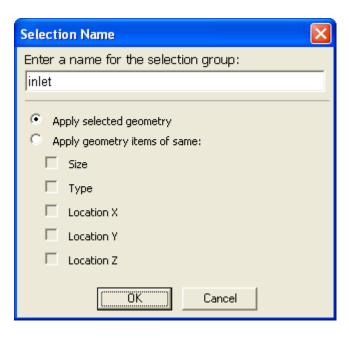
Note that since the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS POLYFLOW, the **Physics Preference** will already be set to **CFD** and the **Solver Preference** will already be set to **POLYFLOW**.

- 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.

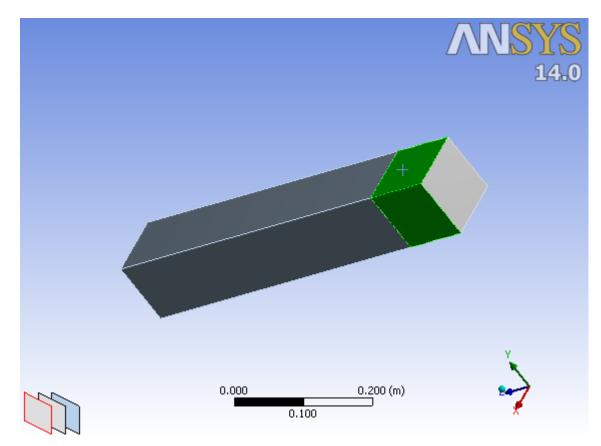


Figure 1.10 The Zero Velocity Faces Selected

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.

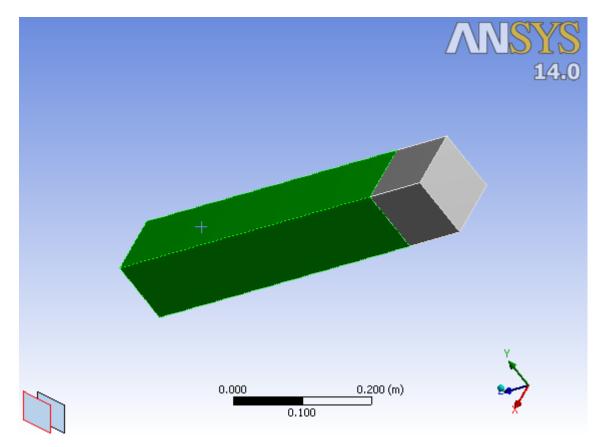
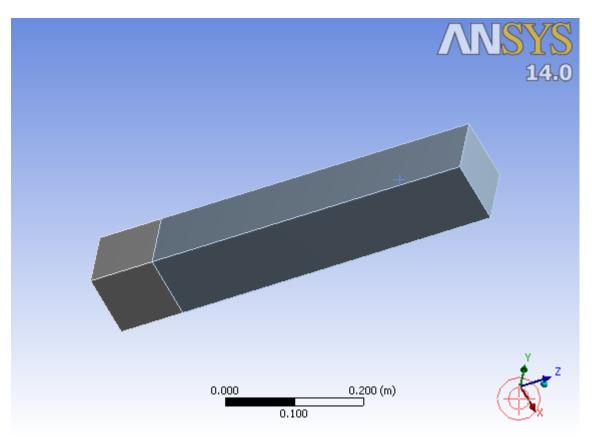


Figure 1.11 The Free Surface Faces Selected

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.

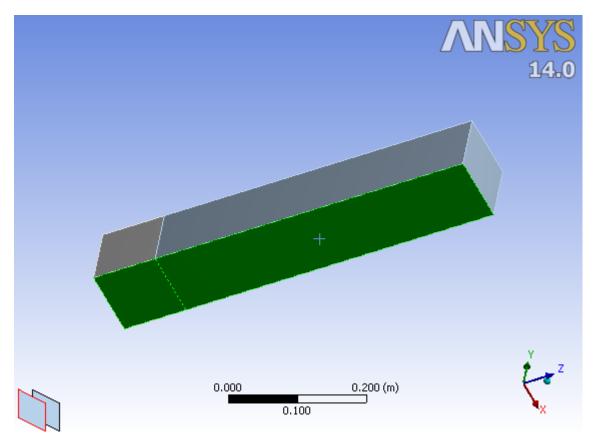


Figure 1.13 The First Pair of Symmetry Faces Selected

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.

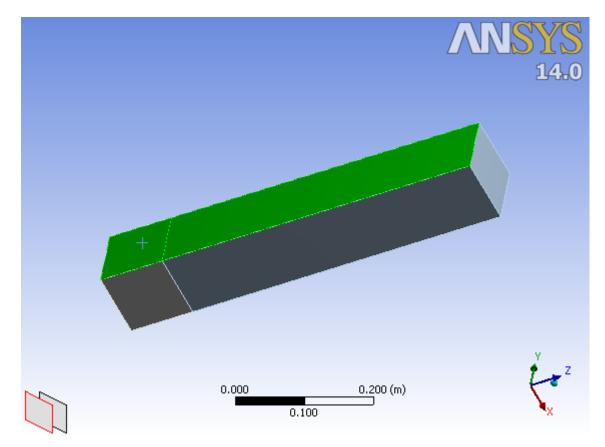
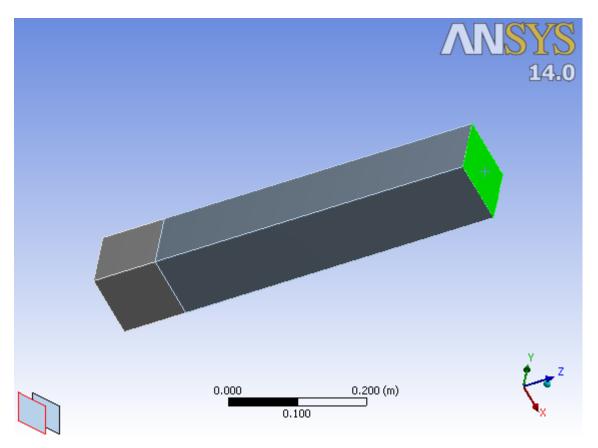


Figure 1.14 The Second Pair of Symmetry Faces Selected

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.

Details of "Mesh"						
Ξ	Defaults		^			
	Physics Preference	CFD				
	Solver Preference	POLYFLOW	_			
	Relevance 0					
	Sizing					
	Use Advanced Size Function	Off 🔽	-			
	Relevance Center	Coarse				
	Element Size	Default				
	Initial Size Seed	Active A				
	Smoothing	Medium	~			

- 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.

De	Details of "Mesh"				
	Defaults		^		
	Physics Preference	CFD			
	Solver Preference	POLYFLOW			
	Relevance 80				
	Sizing				
	Use Advanced Size Function	Off	-		
	Relevance Center	Coarse			
	Element Size Default				
	Initial Size Seed	Active A			
	Smoothing	Medium	v		

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 **Stat-istics** node in the **Details** view to reveal information about the number of nodes, the number of elements, and other details.

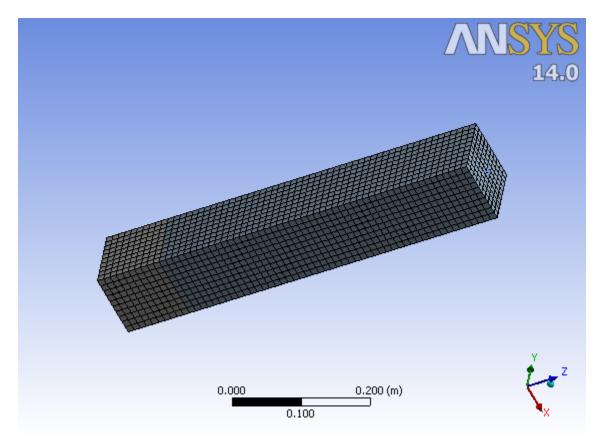
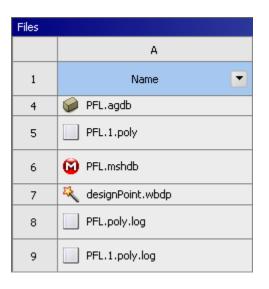


Figure 1.16 The Computational Mesh for the Extrusion Geometry

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**.



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.

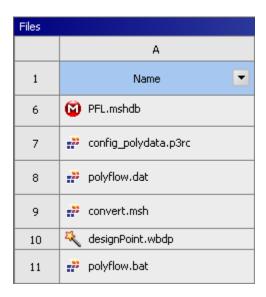
- a. Right-click the **Setup** cell in the ext3d fluid flow analysis system, and click **Import POLYFLOW Dat** ... in the context menu that opens.
- b. Use the **Open** dialog box to browse to the ext3d-workbench folder you unzipped in a previous step, select polyflow.dat, and click **Open**.

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**.



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.

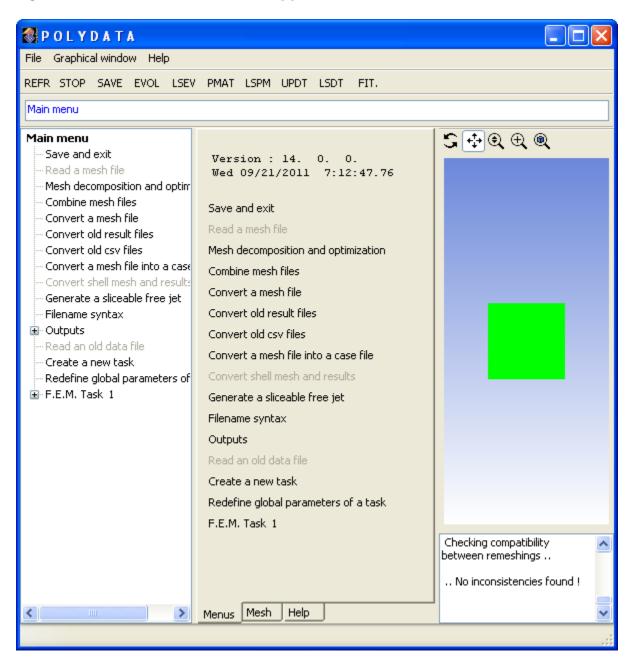
Extra

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.

\overline 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.

Ereate 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.



c. Click 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	
	А
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	nast_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.

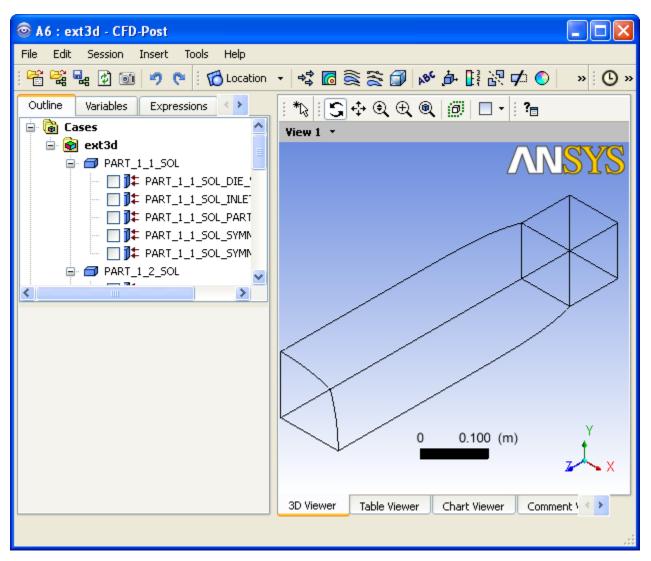
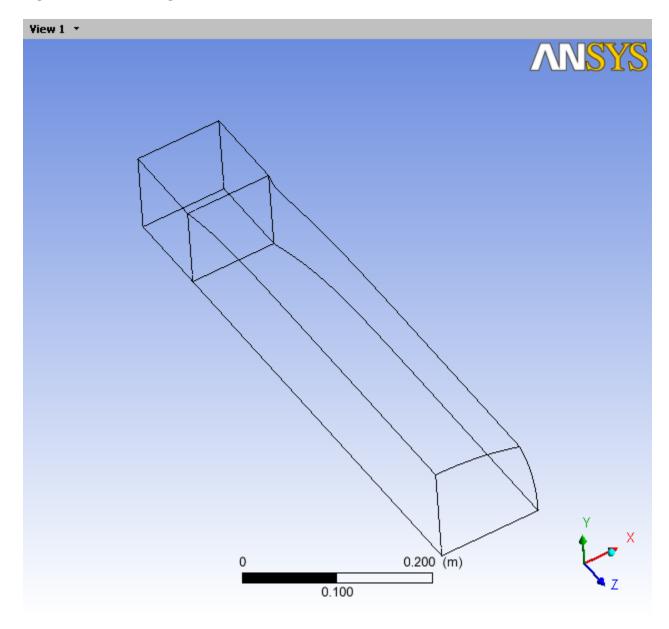


Figure 1.18 The Extrusion Geometry Loaded into ANSYS CFD-Post

- 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.



- 3. Display contours of velocity magnitude on the boundaries (*Figure 1.20* (p. 32)).
 - a. Open the **Insert Contour** dialog box.

Figure 1.19 Rotating the View



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.

Details of Contour 1			
Geometry	Labels Render View		
Domains	All Domains 🛛 🖌 🛄		
Locations	PART_1_1_SOL_DIE_WALL,PAR		
Variable	VELOCITIES		
Range	Global 🔽		
Min	0 [m s^-1]		
Max	0.00207777 [m s^-1]		
Boundary Da	ata 🔘 Hybrid 💿 Conservative		
Color Scale	Linear 🔽		
Color Map	Default (Rainbow) 💌 🖪		
# of Contours	; 11		
📃 Clip to Ra	nge		
Apply	Reset Defaults		

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

💿 Location Selector 🛛 🔀			
😑 ext3d]		
	_SOL_DIE_WALL		
1 PART_1_1	_SOL_INLET		
2 PART_1_1	_SOL_PART_1_2_SOL		
] PART_1_1	_SOL_SYMMETRY_1		
-]‡ PART_1_1	_SOL_SYMMETRY_2		
	_SOL_FREE_SURFACE		
]‡ PART_1_2	_SOL_OUTLET		
	_SOL_PART_1_1_SOL		
	_SOL_SYMMETRY_1		
	_SOL_SYMMETRY_2		
😟 Regions			
ОК	Cancel		

- 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.

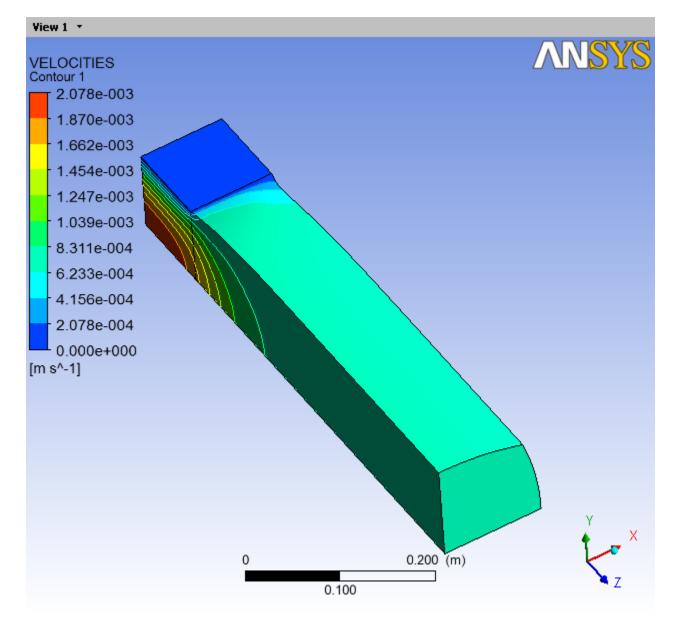
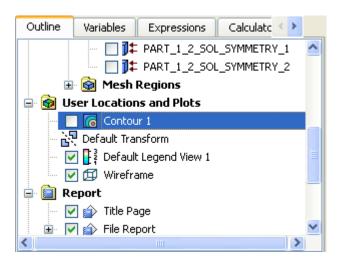


Figure 1.20 Contours of Velocity Magnitude

- 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.

🎯 A6 : ext3d - CFD-Post	
File Edit Session Insert Tool	ls Help
: 😤 🕰 堤 😰 🔟 🤊 🍖 :	🚺 Location 🕝 🤹 🐻 📚 🍣
Outline Variables Expression	Point Cloud
···· □]‡ PART_1_2_ ⊡ Mesh Regions	🗐 Plane
User Locations and Plots User Locations and Plots User Locations and Plots User Locations and Plots Contour 1 User Default Transform User Default Legend View User	isosurface
Details of Contour 1 Geometry Labels Render	 Turbo Surface Turbo Line
Domains All Domains	✓ …

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

💿 Ins	ert Plane	? 🛛
Name	Plane 1	
	ок	Cancel

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

Details of Plar	ne 1	
Geometry	Color Render View	
Domains	All Domains	✓ …
-Definition-		
Method	XY Plane	~
z	0.0 [m]	
-Plane Bound	ds	
Туре	None	~
-Plane Type		
 Slice 	🔘 Sample	
Apply	Reset	Defaults

- 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					
💿 Ins	ert Contour	? 🔀			
Name	Contour 2				
	ОК	Cancel			

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.

Details of Con	tour 2
Geometry	Labels Render View
Domains	All Domains 💌 🛄
Locations	Plane 1,Plane 2,Plane 3,Plane 4 🖌 🛄
Variable	VELOCITIES 💽
Range	Global
Min	0 [m s^-1]
Max	0.00207777 [m s^-1]
Boundary Da	ata 🔿 Hybrid 💿 Conservative
Color Scale	Linear 💙
Color Map	Default (Rainbow) 💌 📳
# of Contour:	s 11 🗘
🔲 Clip to Ra	inge
Apply	Reset Defaults

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

🐵 Location Selector 🛛 🔀
<pre> • ext3d • J* PART_1_1_SOL_DIE_WALL • J* PART_1_1_SOL_JINLET • J* PART_1_1_SOL_SYMMETRY_1 • J* PART_1_2_SOL_SYMMETRY_2 • J* PART_1_2_SOL_OUTLET • PART_1_2_SOL_SYMMETRY_1 • J* PART_1_2_SOL_SYMMETRY_2 • Regions • User Locations and Plots • Plane 1 Plane 2 Plane 3 Plane 4</pre>
OK Cancel

- 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.

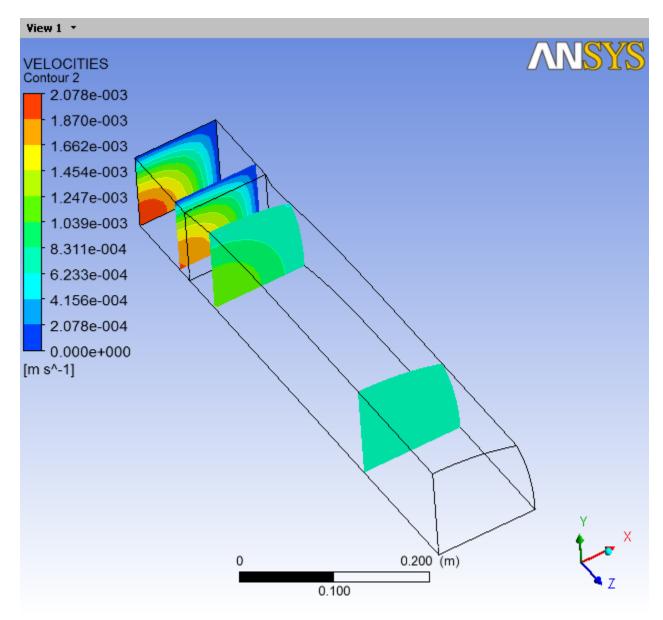
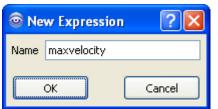


Figure 1.21 Velocity Profiles at Cross Sections

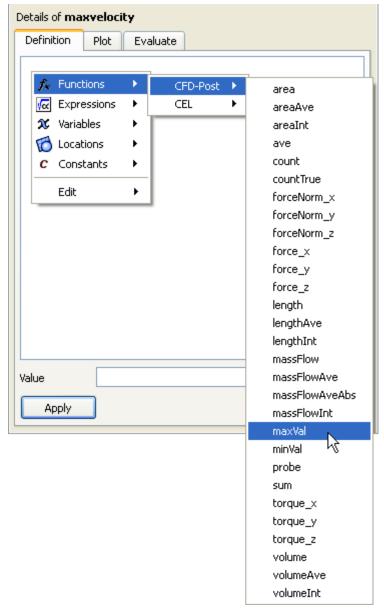
- 5. Create an expression for the maximum velocity at the flow exit, which can be used as an output parameter in ANSYS Workbench.
 - a. Click the **Expressions** tab in the **Tree** view.
 - b. Right-click anywhere in the **Expressions** tab and click **New** ... in the menu that opens to create a new 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.

Definition Plot Evaluate maxVal()@			y	axvelocit	ails of m	Details
Image: Second state sta			Evaluate	Plot	efinition	Defir
Image: Constants Image: Constants Area Image: Constants Image: Constants Locations Image: Edit Image: Constants Normal Image: Normal X Normal X				Ď	ax¥al()	max
X Variables Area Image: Constants LOCAL SHEAR RATE Edit Normal Normal X			•	unctions	f _x	
Constants LOCAL SHEAR RATE Constants Length Edit Normal	_					
C Constants Length Normal Edit Normal X						
Edit Normal X					~	
Edit Normal X			•	Constants	С	
			•	Edit		
Normal Y		Normal X			_	
Normal Z						
PRESSURE						
VELOCITIES		VELOCITIES				
VELOCITIES X		VELOCITIES X K				
VELOCITIES Y		VELOCITIES Y				
VELOCITIES Z		VELOCITIES Z				
Value					Je	Value
				_		_
Apply Y Reset	Reset				Apply	
Z		_				

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.

Details of maxvelocity	
Definition Plot Eva	aluate
max¥al(VELOCITIES)@	
f _∞ Functions → √∞ Expressions → 𝔅 Variables →	
C Constants	PART_1_1_SOL_DIE_WALL PART_1_1_SOL_INLET
Value Apply	PART_1_2_SOL_OUTLET PART_1_2_SOL_PART_1_1_SOL PART_1_2_SOL_SYMMETRY_1 PART_1_2_SOL_SYMMETRY_2 Plane 1 Plane 2 Plane 3 Plane 4 Primitive3d Primitive2d

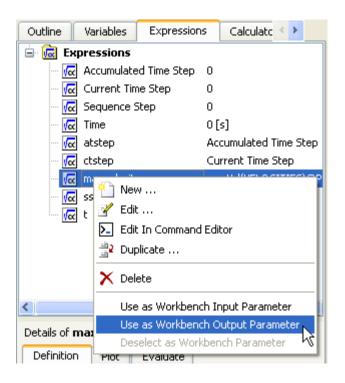
f. Click Apply.

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 \times 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).

Outline	Variables	Expressions	Calculate < 🕨
99	Time atstep ctstep maxvelocity sstep	ne Step 0 Step 0 0 [Ac Cu y ma	cumulated Time Step rrent Time Step xVal(VELOCITIES)@P quence Step
<	maxvelocit	•	
Definitio	_	, Evaluate	
max¥al	(VELOCITIES))@PART_1_2_5	OL_OUTLET
Value		0.0007803	64 [m s^-1]
Арр	ly 🔰		Reset

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 \rightarrow 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

Files	
	А
1	Name 💽
16	xml.xml
17	xml.txt
18	🥯 ext3d.cst
19	🌂 designPoint.wbdp

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

Parameter Charts Parameters Parallel Chart (all) Parameters Chart	1 2 3 4	A ID Input Parameters III (2) ext3d (A1)	B Parameter Name	C Value	1	A Name 💌	B P1 - Row rate	•	C P2 - manvelocity	•
	2	Input Parameters	Parameter Name	Value	1 2	Name 💌	P1 - Row rate			•
Parameters Chart	3				Z					
		🖃 🔯 ext3d (A1)							m s^-1	
					3	Carrent	10		0.00078036	1
		G P1	flow rate	10	•					1
		🚯 New input parameter	New name	New expression						
	6	Output Parameters								
	7	🗏 💽 ext3d(A1)								
	8	P2	manyelocty	0.00070036						
	•	P New output		New expression						
	10	Charts								
	<				c				_	
	REPORT	ies: No data		* ± X	Overt					
		A	8							
	1	Property	Value							

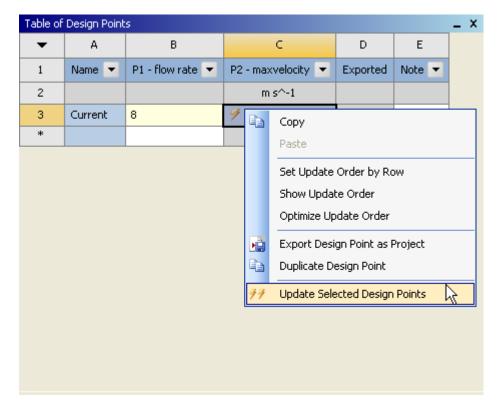
- 2. Run the calculation again with a new inlet flow rate for the current design point.
 - a. Enter 8 under P1 flow rate for the Current design point (i.e., cell B3) in the Table of Design Points.

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.

Table of	Design Points	5			⊸ д	×
	А	В	с	D	E	
1	Name 💌	P1 - flow rate 🛛 💌	P2 - maxvelocity 💌	Exported	Note 💌	
2			m s^-1			
3	Current	8	0.00062376			
*						

- 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.

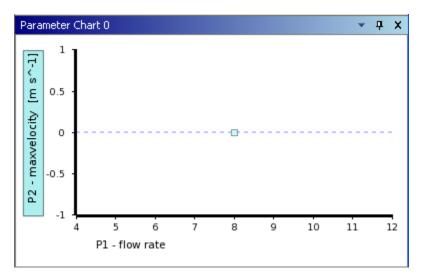
The **Properties of Outline A11:0** window will display an initial setup for **Parameter Chart 0**, in which **P1 - flow rate** is selected from the **X-Axis (Bottom)** drop-down list.

Propertie	es of Outline A11: 0	<u>~</u> џ Х
	А	В
1	Property	Value
2	Parameter Chart: General	
3	Exclude Current Design point	
4	X-Axis (Bottom)	P1 - flow rate 💌
5	X-Axis (Top)	•
6	Y-Axis (Left)	P2 - maxvelocity
7	Y-Axis (Right)	•

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	Table of Design Points 🗾 👻 📮 🗙					×
	А	В	с	D	E	
1	Name 💌	P1 - flow rate 🛛 💌	P2 - maxvelocity 💌	Exported	Note 💌	
2			m s^-1			
3	Current	8	0.00062376			
4	DP 1	10	7			
*						

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., $ext3d-wb_dpl.wbpj$) 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.

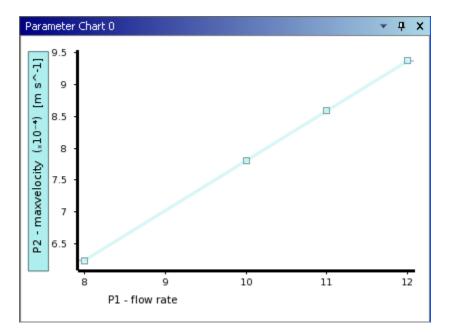
Click Update All Design Points in the ANSYS Workbench toolbar.

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.

Table of Design Points 🔹 🕂 🗙					×	
	А	В	с	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 1.26 Displaying Values for All of the Design Points

Figure 1.27 The Chart of All of the Design Points



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

$\textbf{File} \rightarrow \textbf{Save}$

- 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

Files	
	А
1	Name 💌
15	📅 res
16	🥯 ext3d.cst
17	🌂 designPoint.wbdp
18	TempWmicBatchFile.bat

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.