

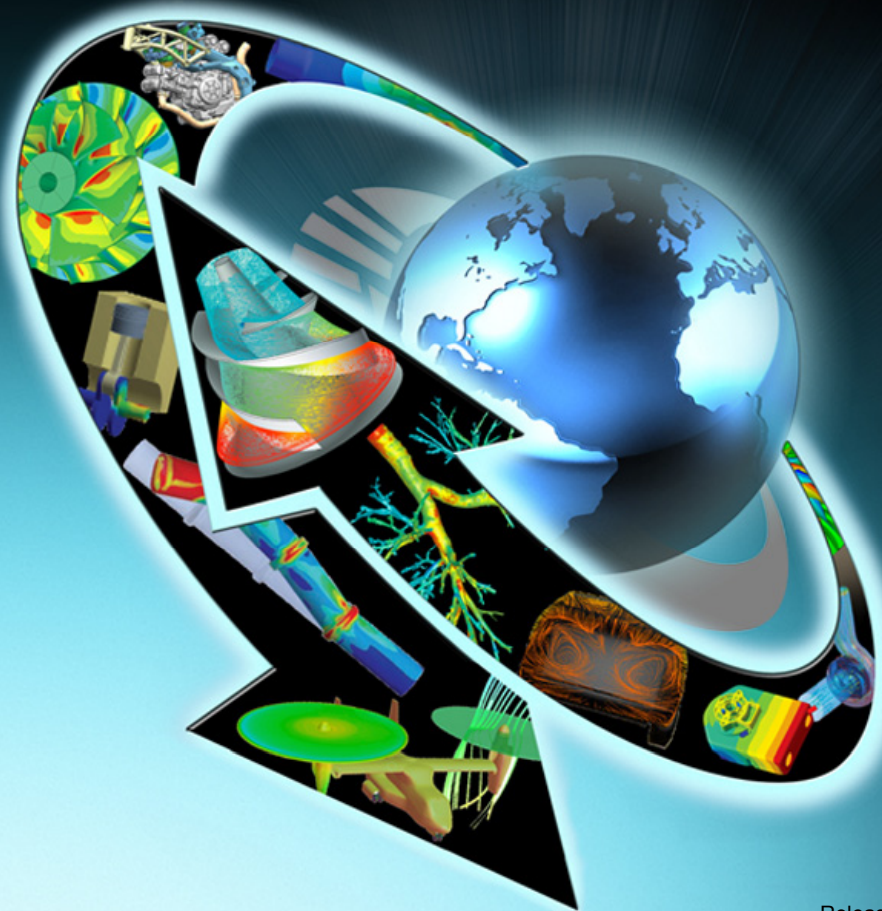


Customer Training Material

Lecture 1

Welcome

Introduction to ANSYS FLUENT



- Welcome to the **ANSYS FLUENT** introductory training course!
- This training course covers the basics of using ANSYS FLUENT for performing a range of Computational Fluid Dynamic (CFD) analyses.
- It is intended for all new or occasional ANSYS FLUENT users
- **Course Contents:**
 - General understanding of the CFD process, and use of the FLUENT interface
 - Setting up a CFD simulation (materials, boundary conditions etc.)
 - Ensuring flow accuracy
 - Accounting for Turbulence and Heat Transfer
 - Basics of User Defined Functions (UDFs)
 - Basics of Multiphase, Combustion and Dynamic Mesh
 - Postprocessing

Introductions:

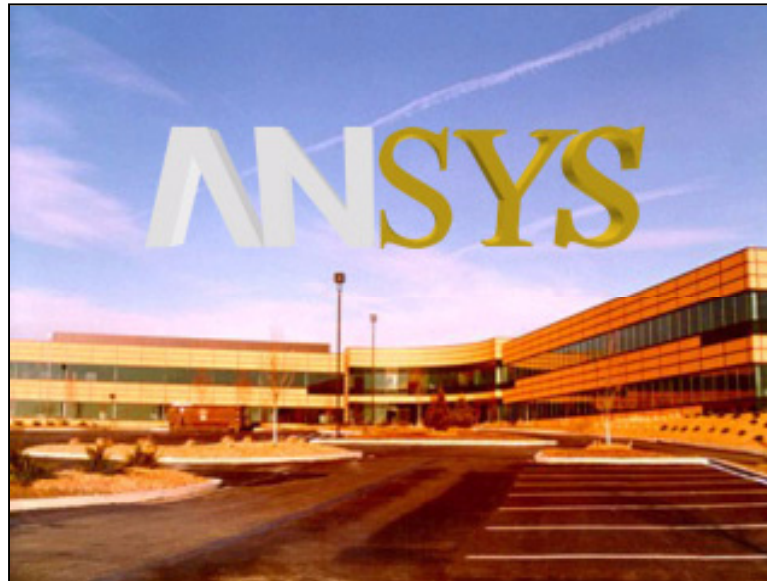
- **Introducing your trainer**
- **Class introductions**
 - Please share:
 - Your past experience (if any) of using CFD
 - The types of simulation you wish to perform (aerodynamics, combustion, free-surface flows etc).
- **Local Information**
 - Fire evacuation
 - Location of Toilets / Restrooms
 - Refreshments
 - Smoking policy
 - Any other local details

- 8:30 – 9:00 Lecture 1: Introduction and Welcome
- 9:00 – 9:45 Lecture 2: Introduction to CFD Methodology / Demo
- 9:45 – 10:30 Workshop 1 (Mixing Tee)
- 10:30 – 10:45 Break
- 10:45 – 11:15 Lecture 3: Basic Overview of Using FLUENT Interface
- 11:15 – 12:00 Workshop 2 (Airfoil)
- 12:00 – 1:00 Lunch
- 1:00 – 1:45 Lecture 4: Cell Zones and Boundary Conditions
- 1:45 – 2:45 Workshop 3 (Multi-Species Flow)
- 2:45 – 3:00 Break
- 3:00 – 4:00 Lecture 5: Solver Settings
- 4:00 – 5:00 Workshop 4 (Electronics Cooling)

8:30 – 9:30	Lecture 6: Turbulence Modeling
9:30 - 10:15	Workshop 5 (MRF and Sliding Meshes)
10:15 – 10:30	Break
10:30 – 11:15	Lecture 7: Heat Transfer
11:15 – 12:00	Workshop 6 (Discrete Phase Model)
12:00 – 1:00	Lunch
1:00 – 1:45	Lecture 8: User Defined Functions (UDFs)
1:45 – 2:30	Workshop 7 (Tank Flush)
2:30 – 3:30	Lecture 9: Advanced Physics / Workshop
3:30 – 3:45	Break
3:45 – 4:15	Lecture 10: Transient Flow Modeling
4:15 – 5:00	Lecture 11: Postprocessing / Workshop

ANSYS, Inc.

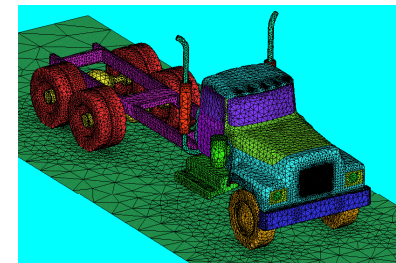
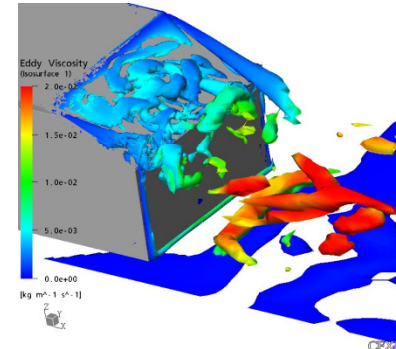
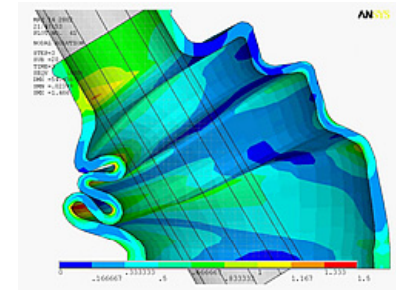
- Developer of ANSYS family of products
- Global Headquarters in Canonsburg, PA - USA (south of Pittsburgh)
 - Development and sales offices in U.S. and around the world
 - Publicly traded on NASDAQ stock exchange under “ANSS”
 - For additional company information as well as descriptions and schedules for other training courses visit www.ansys.com



... ANSYS Family of Products

ANSYS, Inc. Family of Products include the following:

- **ANSYS Workbench Applications** – A development framework upon which ANSYS applications are built. Applications are available in the areas of structural mechanics, geometry, CFD, rigid dynamics, electromagnetics, optimization and more.
- **Mechanical APDL** – traditional ANSYS command driven interface.
- **ANSYS CFD** – State-of-the-art CFD solvers, including CFX and FLUENT.
- **ANSYS ICEM CFD** – Application containing CFD meshing tools with general pre- and post-processing features.
- **ANSYS AUTODYN** – Explicit dynamic solver for transient non-linear simulations involving large deformations and strains, non-linear material behavior, non-linear buckling, complex contact, fragmentation, and shock wave propagation.
- **ANSYS LS-DYNA** – LSTC's LS-DYNA explicit dynamic solver technology with the pre-/post-processing power of ANSYS software. This powerful pairing can be used to simulate crash tests, metal forging, stamping, and catastrophic failures.



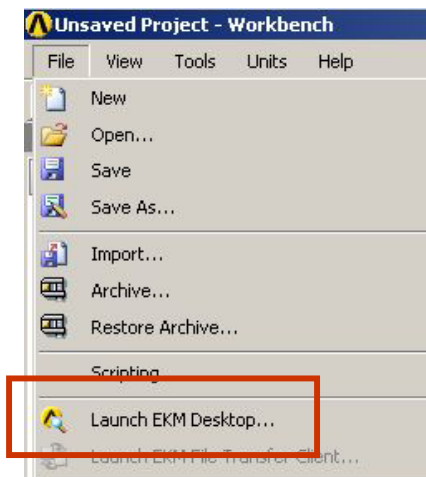
- **What is ANSYS EKM?**

- ANSYS Engineering Knowledge Manager (EKM) is a web-based multi-user collaborative product that is aimed at meeting the simulation data and process management challenges faced by our customers. EKM is tightly integrated with other ANSYS simulation offerings and it can also be very easily integrated with other simulation codes including legacy and competitor's software.

- **What is ANSYS EKM Desktop?**

- ANSYS EKM Desktop is a single user, local environment version of EKM. It is available as part of ANSYS R13 release and it can be accessed via Workbench. EKM Desktop is focused at meeting the challenge of “re-using existing simulations” and thereby increasing simulation productivity and efficiency.

- **EKM Desktop can be started under the File menu in the Project window**



- This document provides quick tour of the ANSYS Customer Portal, including:
 - Using the [Software Download](#) area
 - Product information
 - Customer Support & Training tools
 - Knowledge Resources / FAQs
 - Training Material & Animated Tutorials
 - [Online Support Request](#)
 - Phone number
 - Other tools

Introduction to ANSYS FLUENT

ANSYS Customer Portal

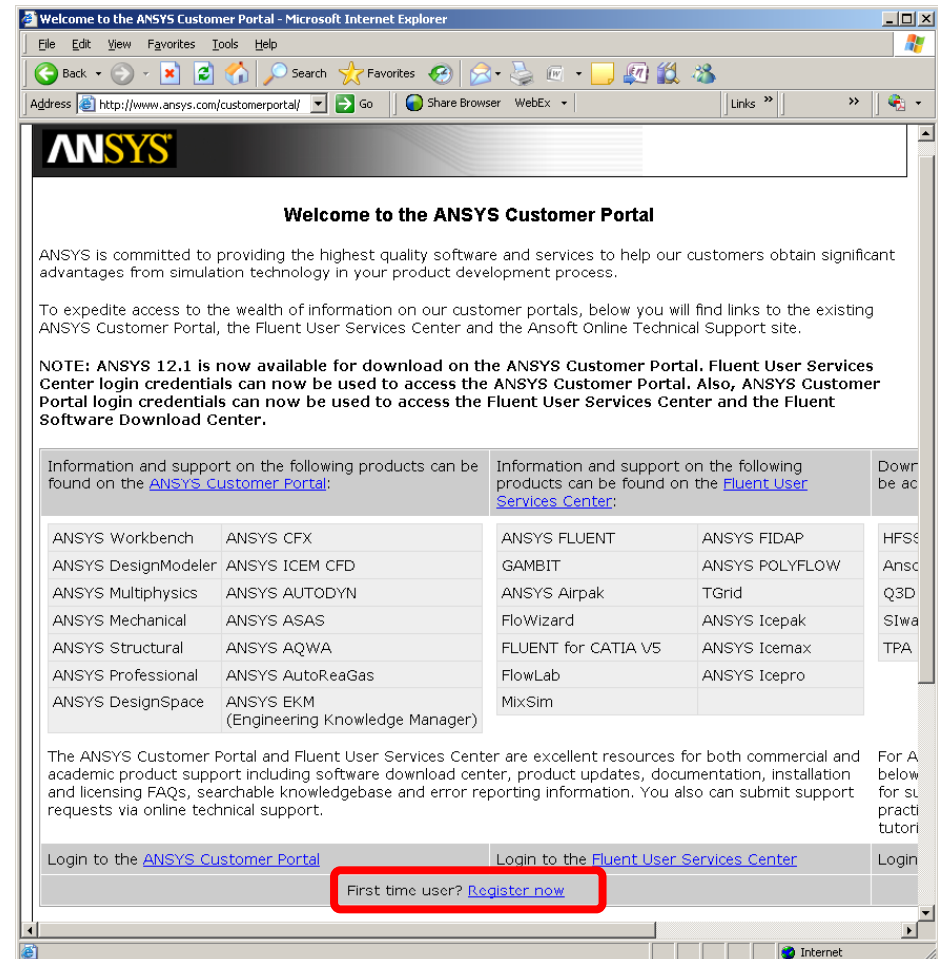


1. Visit <http://www.ansys.com/customerportal/>
2. For first time users, click [Register now](#)
3. For returning users, click one of the following:
 - [ANSYS Customer Portal](#) for Mechanical/Fluent/CFX/Ansoft Support
 - [Fluent User Services Center](#) for legacy Fluent downloads

Notes:

- If you are encountering difficulties with the ANSYS Customer Portal please click on the [ANSYS Portal Help Support Request Form](#) on the login page
- Online Technical Support for all ANSYS products, Mechanical/Fluent/CFX/Ansoft, is available through the [ANSYS Customer Portal](#)

<http://www.ansys.com/customerportal/>



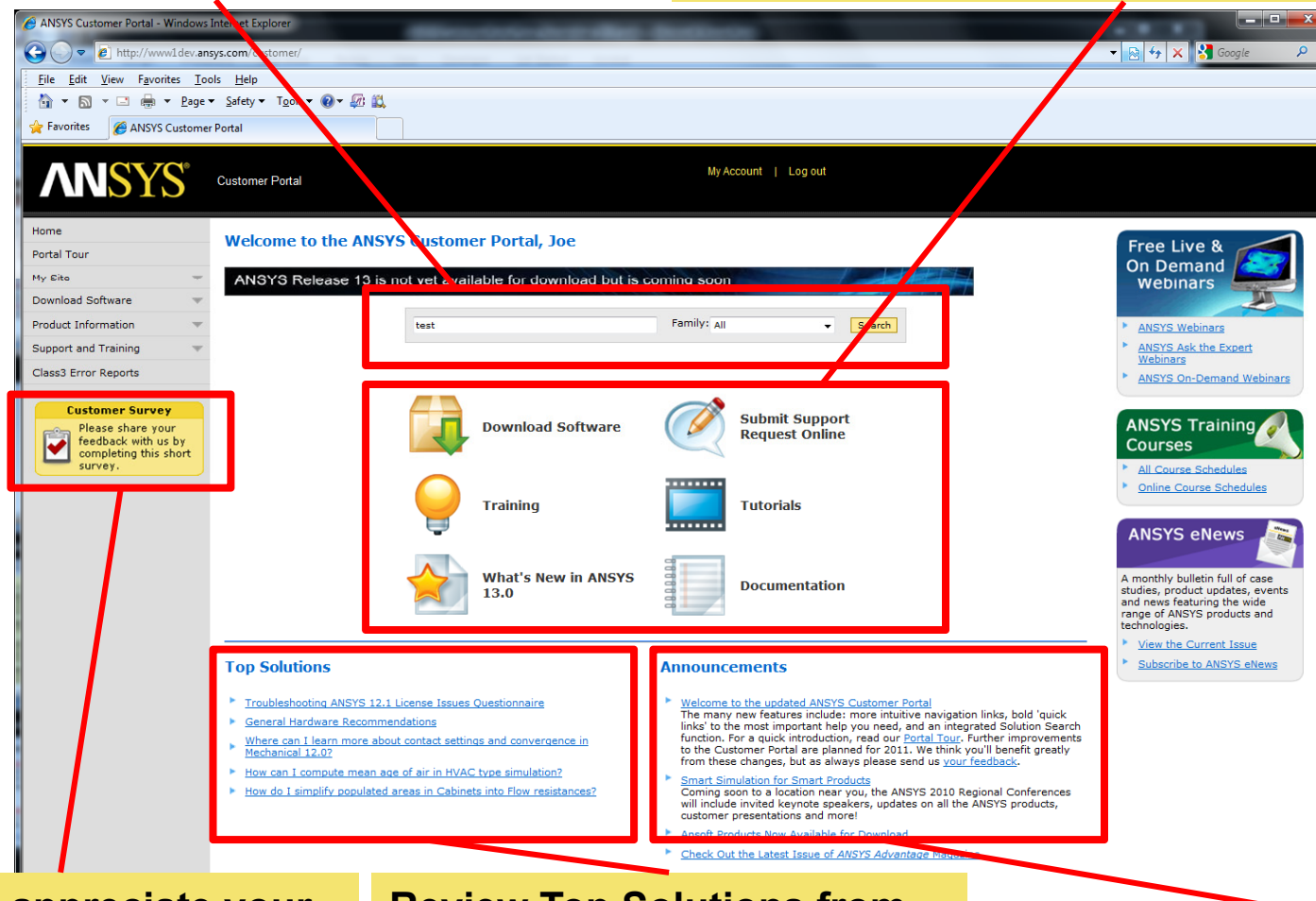
Introduction to ANSYS FLUENT

ANSYS Customer Portal Home Page



Search the Knowledge Resources for FAQs, Best Practices, Examples!

Navigate to important locations on the Portal with the click of an icon!



Survey: We appreciate your feedback!

Review Top Solutions from our Knowledge Resources!

Review latest News!

Introduction to ANSYS FLUENT Support and Training

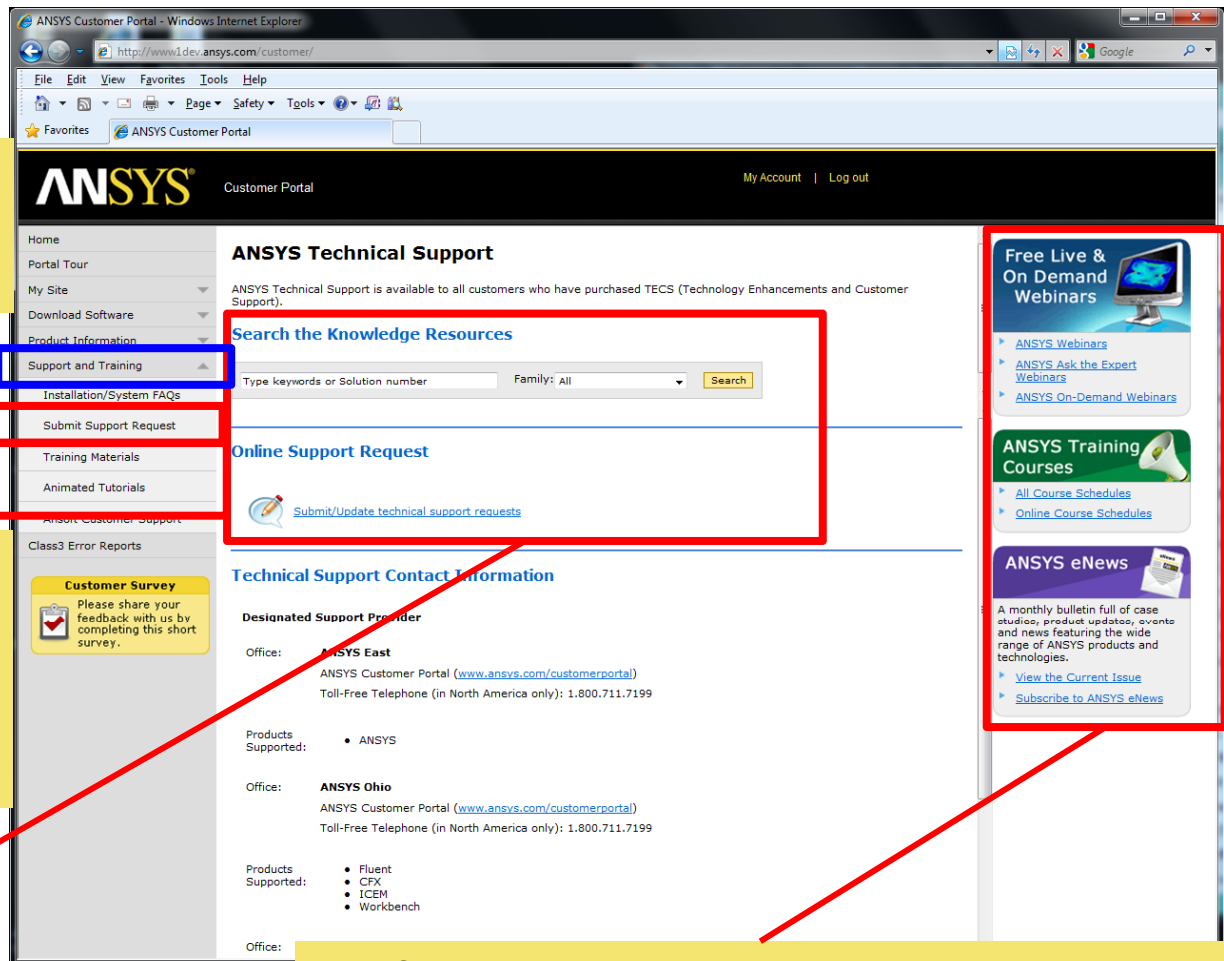


Support Request
Submit a question to a support specialist.

Animated Tutorials & Training Materials
Watch short 2-8 minute demo's
Download PDF versions of the training lectures

Search KRs & Submit SRs
Type keywords to perform a search
Submit/Update support requests

Useful Links
Register for live technical webinars
Register for training courses
Register/view the ANSYS eNews



For ANSYS Direct Customers*:

From the ANSYS Customer Portal

1. Click **Submit Support Request**
2. Click **Submit/Update technical support request**
3. Under *My Service Requests*, click **New**

Other Methods

For North America:

- Phone **1-800-711-7199** (make sure that your Contact # (aka. Individual Customer #) is available when you call!)
- Support hours – 7:30am to 8:00pm Eastern time

*** Customers serviced by ANSYS Channel Partners should contact that Channel Partner for technical support**

Check the Status of Existing Support Requests



For ANSYS Direct Customers*:

From the ANSYS Customer Portal

1. Click **Submit Support Request**
2. Click **Submit/Update technical support request**
3. Click the **SR#** of interest

Through Phone in North America

1. Call **1-800-711-7199**
2. Press **1** for Technical Support
3. Press **2** for Existing Technical Support
4. Enter your **SR number** followed by **#** to be connected to the support specialist working on your request

SR #	Status	Opened
15019031	Closed	7/14/2009 04
14599125	Closed	7/8/2009 04
14599121	Closed	7/8/2009 04
13662521	Closed	6/28/2009 04
13662516	Closed	6/28/2009 04
13557721	Closed	6/26/2009 14

*** Customers serviced by ANSYS Channel Partners should contact that Channel Partner to check on status of existing support requests**

Attachments

Upload large files (>20 MB) to the ANSYS FTP. Attach small files (<20 MB) directly to SR. [Note: To download attachments you may need to add <https://crm.ansys.com> to your list of trusted sites.]

ITAR Data

Please do not attach ITAR controlled data to your SR.

SR Updates

Communicate directly with your support specialist by adding & viewing SR Updates.

Associated Solutions

Previously created solutions associated with your SR may be viewed here.

My Service Requests

New Query

SR #	Status	Opened
15019031	Closed	7/14/2009 0
14599125	Closed	7/8/2009 04
14599	Closed	7/8/2009 04
13662	Closed	6/28/2009 0
13662	Closed	6/28/2009 0
13557	Closed	6/26/2009 1

Service Request

Menu

SR #: 15019031
Status: Closed
Account: MCM Technology
Contact: Joe Customer
Assigned To: Jim Pasquerelli
Summary: test system calls
Opened: 7/14/2009 09:11:02 AM
Closed: 7/14/2009 02:48:40 PM
Description: test system calls

Family: Structural Mechanics
Application: Mechanical (f. Simulation)
Area: Systems/Installation
SubArea: General
Product Version: 11.0
Platform: AMD/Intel 32-bit
OS: AIX 5.3
Priority:

Please use ANSYS FTP (<ftp://ftp.ansys.com/incoming/techsupp>) for attachments greater than 20 MB. be removed.

Attachments

Add

Attachment Name	File Type	Size	Date and Time
-----------------	-----------	------	---------------

Please create an Update item when attachments are added or files have been uploaded using FTP

SR Updates

Add

Description	Updated
close call jim	7/14/2009 09:11:53 AM

Associated Solutions

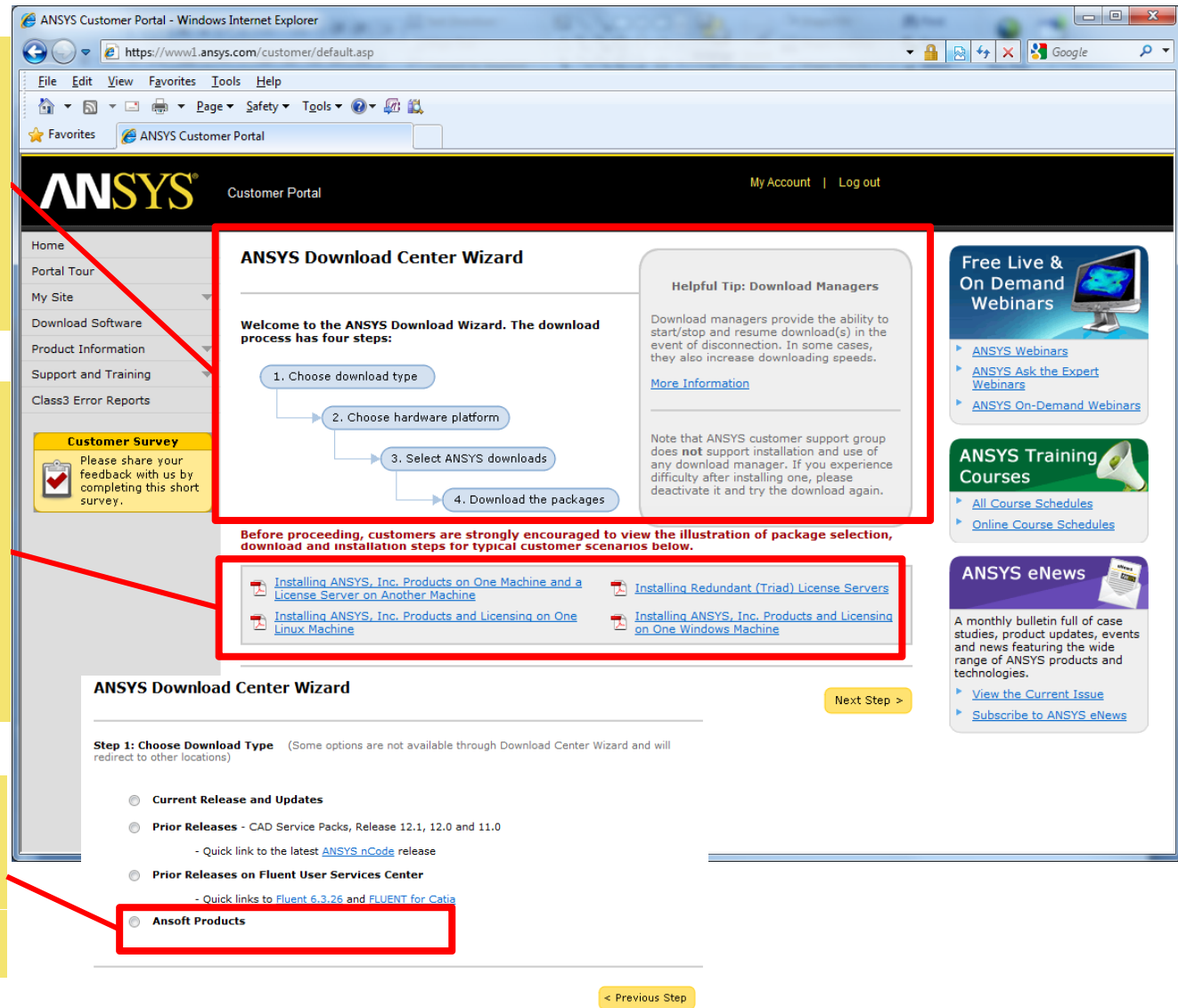
Query

Number	Created
--------	---------

The Download Wizard
Will guide you through 4
easy steps to download
the products you need.
The current release is
ANSYS 13.0!

Installation Tutorials
Download these tutorials
before proceeding. They
include step-by-step
instructions on installing
and setup your
applications

Download Ansoft
Products
Select "Ansoft Products"
on Step 1



ANSYS Customer Portal - Windows Internet Explorer

https://www1.ansys.com/customer/default.asp

File Edit View Favorites Tools Help

Home Portal Tour My Site Download Software Product Information Support and Training Class3 Error Reports

ANSYS Customer Portal

My Account | Log out

ANSYS Download Center Wizard

Welcome to the ANSYS Download Wizard. The download process has four steps:

1. Choose download type
2. Choose hardware platform
3. Select ANSYS downloads
4. Download the packages

Helpful Tip: Download Managers

Download managers provide the ability to start/stop and resume download(s) in the event of disconnection. In some cases, they also increase downloading speeds.

[More Information](#)

Note that ANSYS customer support group does not support installation and use of any download manager. If you experience difficulty after installing one, please deactivate it and try the download again.

Before proceeding, customers are strongly encouraged to view the illustration of package selection, download and installation steps for typical customer scenarios below.

- [Installing ANSYS, Inc. Products on One Machine and a License Server on Another Machine](#)
- [Installing ANSYS, Inc. Products and Licensing on One Linux Machine](#)
- [Installing Redundant \(Triad\) License Servers](#)
- [Installing ANSYS, Inc. Products and Licensing on One Windows Machine](#)

ANSYS Download Center Wizard

Step 1: Choose Download Type (Some options are not available through Download Center Wizard and will redirect to other locations)

- ☐ Current Release and Updates
- ☐ Prior Releases - CAD Service Packs, Release 12.1, 12.0 and 11.0
 - Quick link to the latest [ANSYS nCode](#) release
- ☐ Prior Releases on Fluent User Services Center
 - Quick links to [Fluent 6.3.26](#) and [FLUENT for Catia](#)
- ☒ Ansoft Products

< Previous Step

Next Step >

Customer Survey

Please share your feedback with us by completing this short survey.

Free Live & On Demand Webinars

- [ANSYS Webinars](#)
- [ANSYS Ask the Expert Webinars](#)
- [ANSYS On-Demand Webinars](#)

ANSYS Training Courses

- [All Course Schedules](#)
- [Online Course Schedules](#)

ANSYS eNews

A monthly bulletin full of case studies, product updates, events and news featuring the wide range of ANSYS products and technologies.

- [View the Current Issue](#)
- [Subscribe to ANSYS eNews](#)

Introduction to ANSYS FLUENT

FLUENT User Services Center



Note: The layout of this page may change with further updates to the User Services Center

Product Information

Click on a product to view FAQs, documentation, tutorials, best practices, libraries, etc.

Animated Tutorials

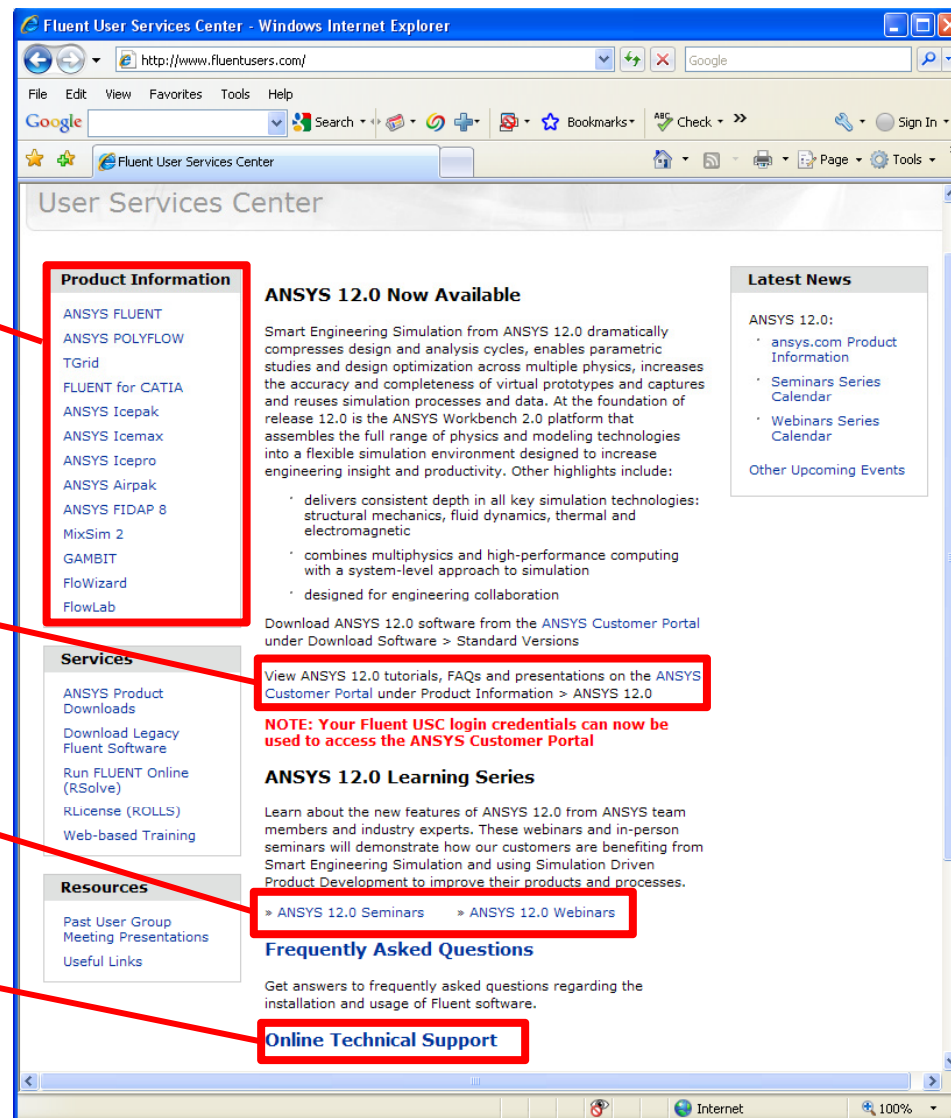
Watch short 2-8 minute demonstrations and then walk through the example files on your own.

Lecture Series

Register to watch live webinars on new features. (1 hour)

Support Request

Submit a question to a support specialist.



**Thank you in advance for using the
ANSYS Customer Portal!**

**Consider filling out the Survey on the
Customer Portal. We appreciate your
feedback, which will help us continue to
improve the services we offer!**

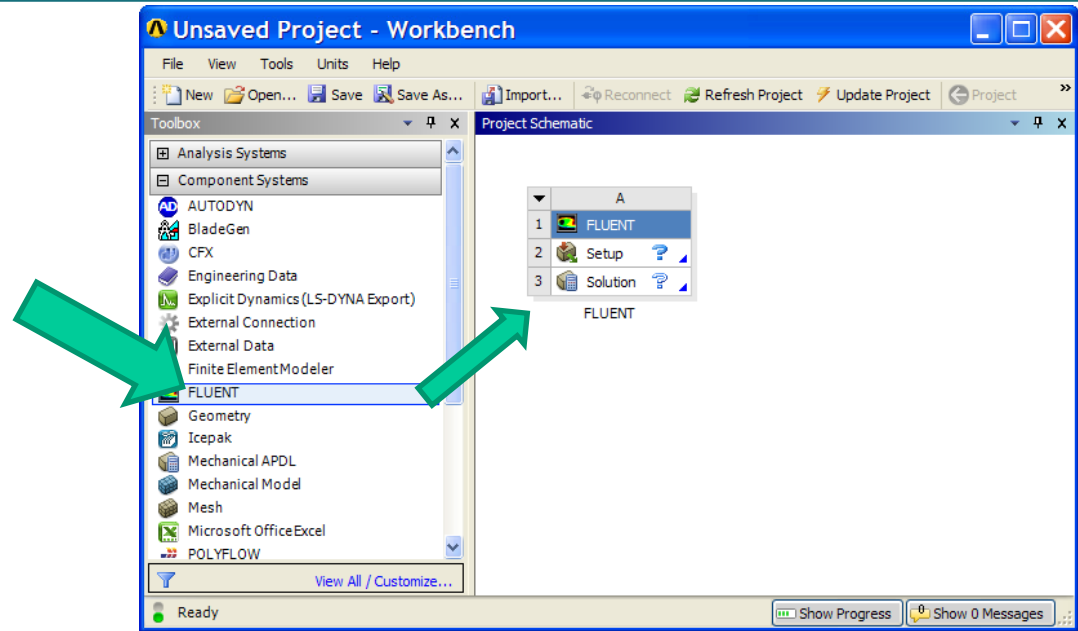
Introduction to ANSYS FLUENT

Starting FLUENT [1]

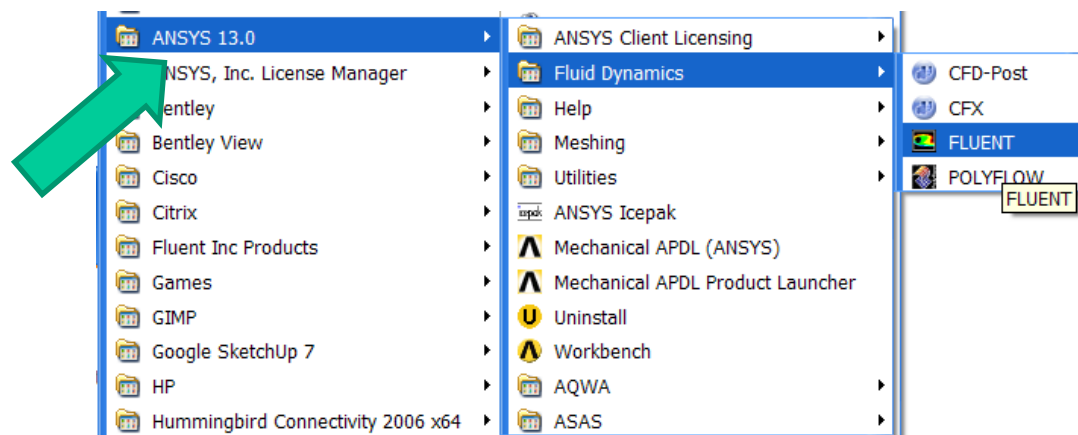


FLUENT may be run either

- From within Workbench, by dragging a FLUENT session into the workspace



- Or from the Start Menu (or Windows / UNIX command prompt, type 'fluent')



Introduction to ANSYS FLUENT

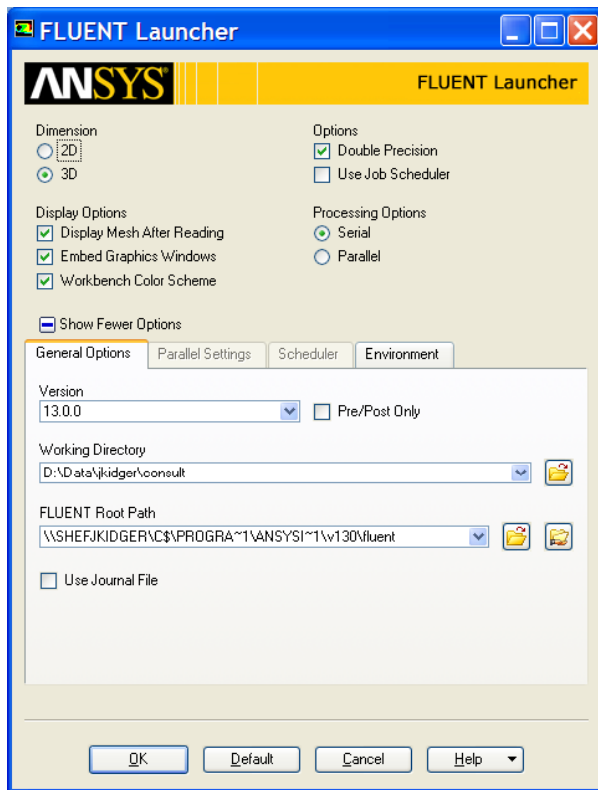
Starting FLUENT [2]



A launching window then appears from which you need to select how FLUENT is to run (serial / parallel) (2D/3D) (single / double precision)



Launcher Screen (Workbench)



If running FLUENT standalone, there are additional settings from where you can set the working directory (or indeed pick different releases if you have those installed)

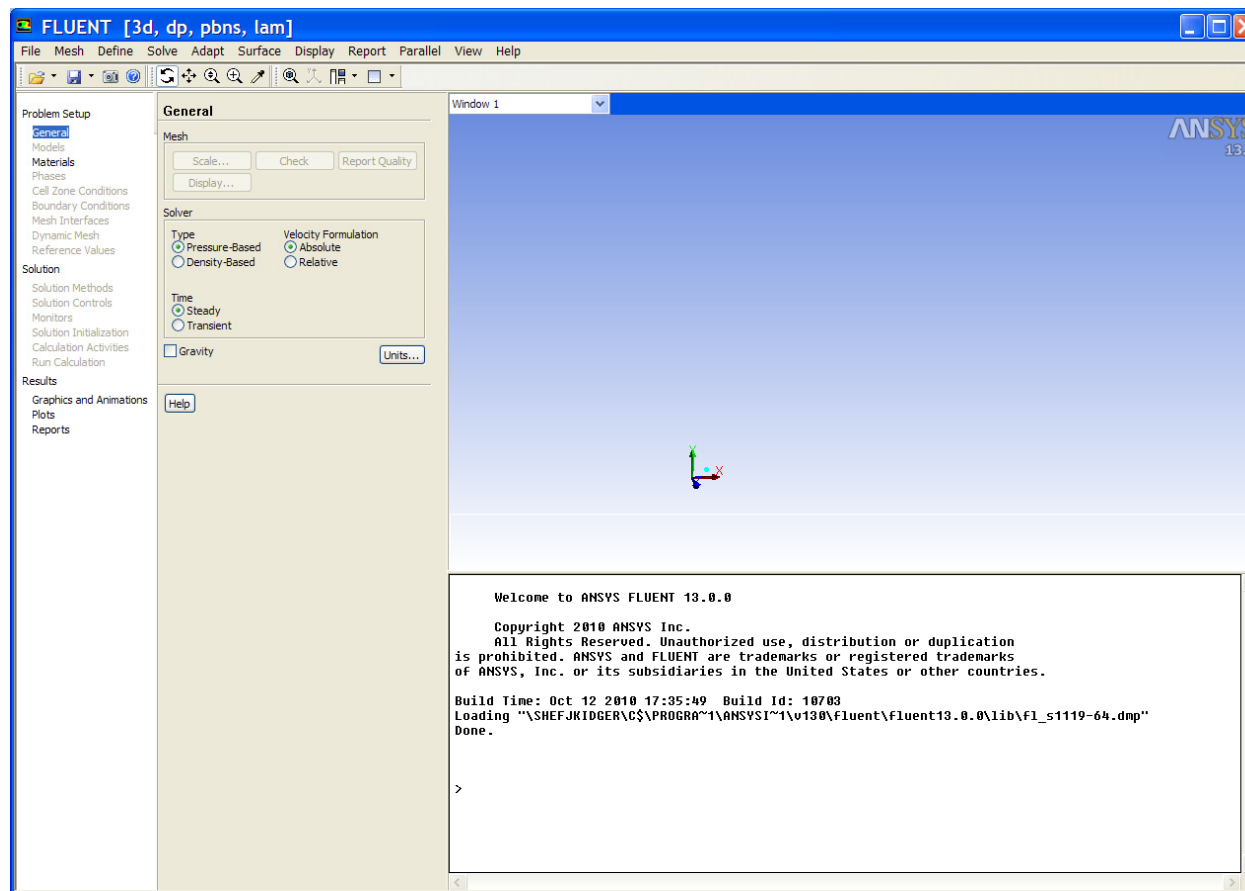
Introduction to ANSYS FLUENT

Starting FLUENT [3]



The FLUENT session will look like this when first started.

Use of the interface will be covered in detail in the subsequent lectures.



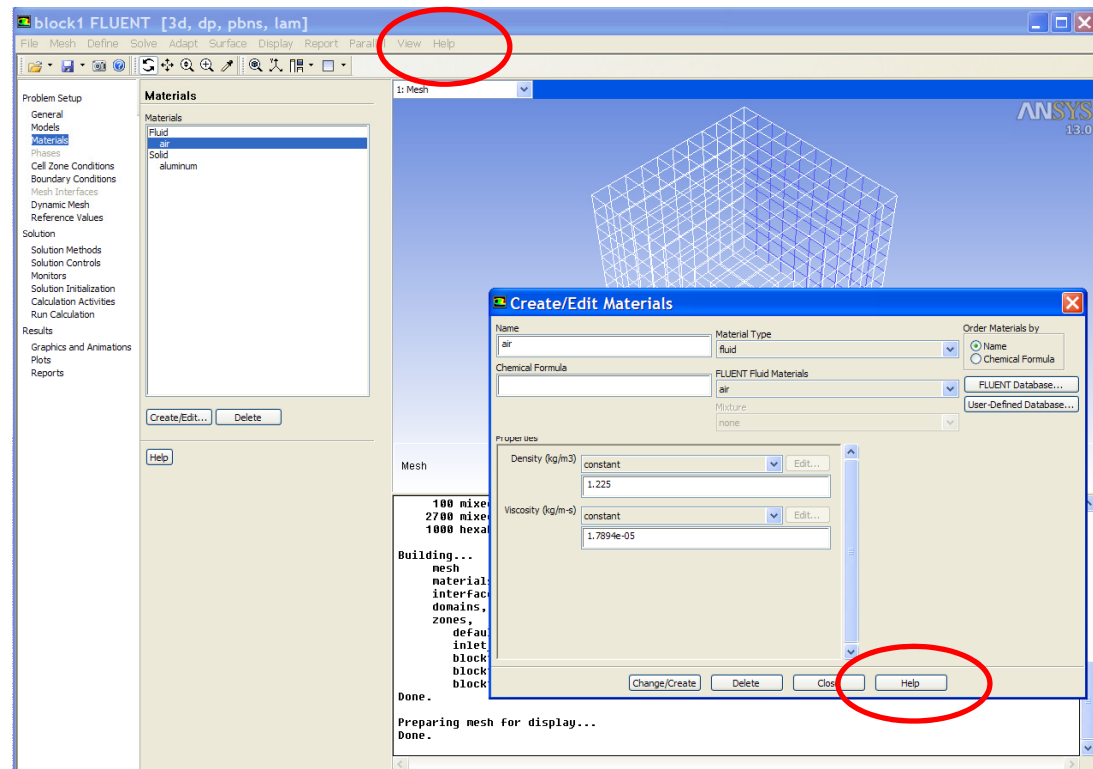
- FLUENT is supplied with comprehensive documentation:
 - **Getting Started Guide** – The basics
 - **User's Guide** – Providing instruction on how to set up simulations and what options are available
 - **FLUENT in Workbench User's Guide** – How FLUENT functions in the Workbench environment
 - **Theory Guide** – Providing detail of the formulae used by the FLUENT solver
 - **UDF Manual** – how to customise FLUENT using your own code
 - **Tutorial Guide** – Contains many more workshops in addition to those covered in this course including advanced and specialist applications (combustion, dynamic mesh, multiphase etc.) Accompanying input files can be downloaded from the customer portal
- Access Help by either:
 - Pressing 'help' or F1 with the FLUENT GUI
 - Start Menu....
 - Looking online in the User Services Centre

Introduction to ANSYS FLUENT

Documentation [2]



- Within FLUENT, every panel has a 'Help' option, or for more general help click on the top toolbar.
- Alternatively, the documentation can be accessed either:
 - On the Customer Portal,
 - Or following **Start Menu > All Programs > ANSYS 13.0 > Help**



Introduction to ANSYS FLUENT Documentation [3]



- Specific topics can be quickly located and accessed using the ANSYS Help contents tree or by using the Search/Index tabs.

