

# Configuring and Launching ANSYS FLUENT 16.0 - Distributed using IBM Platform MPI or Intel MPI

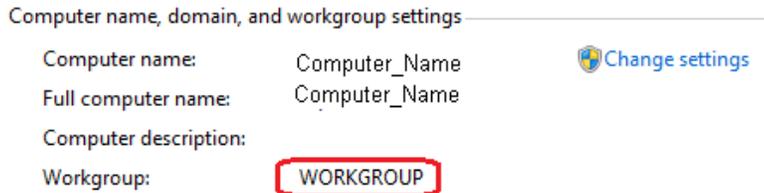
## Table of Contents

BEFORE YOU PROCEED.....	1
Launching FLUENT Using Shared Memory.....	2
Configuring FLUENT to run across Multiple Computers (Distributed Memory).....	2
Sharing a Directory.....	2
Mapping a Network Drive.....	3
Installing the Message Passing Interface (MPI).....	3
Installing IBM Platform MPI or Installing INTEL MPI.....	3
Installing IBM Platform MPI.....	5
Issue We Have Seen and Best Practices.....	8
Launching FLUENT.....	8
Troubleshooting.....	10
Firewalls.....	10

## BEFORE YOU PROCEED

ANSYS Fluent runs using MPI (required for running on more than one computer) must be members of an Active Directory domain and the same Active Directory domain. Trying to run Fluent or any ANSYS solvers requiring MPI on more than one computer and being in a “**WORKGROUP**” environment instead of a **DOMAIN** will **not** work and is not supported by the MPI vendors or ANSYS. Active Directory is used by the MPIs for username/password authentication for security purposes. Before you proceed check your system to make sure you are a member of a Domain. Open up **Control Panel > System**, look for the term “**Domain**” with a domain name. **If you see the word “WORKGOUUP” do not proceed and contact your IT folks about adding your computer to Active Directory.**

1. Go to **Control Panel > System** to check if your computer is a member of a **WORKGROUP** or **DOMAIN**



**NOTE:** The steps listed in this document are for running FLUENT across multiple computers **ONLY** not for running on multiple processors on the same computer. Some of these steps require administrator privileges. Please contact your Systems Administrator for assistance.

## Launching FLUENT Using Shared Memory

To launch FLUENT using multiple processors (Shared Memory) on the same computer select parallel in the FLUENT Launcher and then choose the number of processors. There is no need to install or configure an MPI.

NOTE: If you are just running Fluent on one computer there is no need to proceed further

---

## Configuring FLUENT to run across Multiple Computers (Distributed Memory)

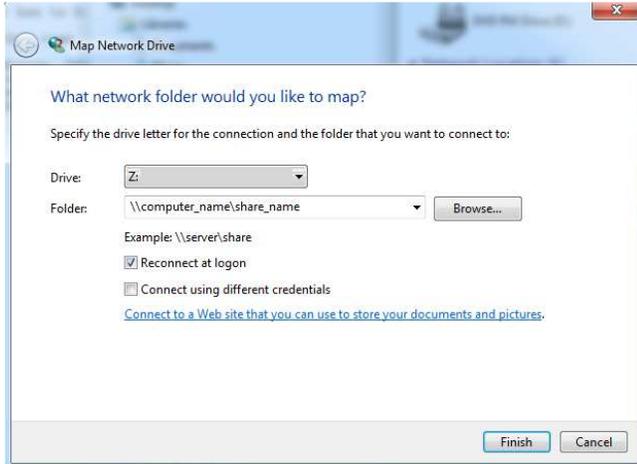
1. Install FLUENT (it is only necessary to install FLUENT on the submit machine). The default location will install FLUENT in C:\Program Files\ANSYS Inc\v160\fluent
2. Share the ANSYS Inc directory located by default at C:\Program Files\ANSYS Inc o that all computers on the cluster can access this shared directory through the network. Make sure you give "**Full Control**".

### Sharing a Directory

1. Right-click on the **ANSYS Inc** directory located at C:\Program Files\ANSYS Inc
2. Go to **Properties (Not Share With)**
3. Choose the "**Sharing**" tab > **Advanced Sharing**.
4. Check "**Share this Folder**"
5. Select the **Permissions** button
6. Choose "**Full Control**"
7. Create a working directory on any computer that you will be launching Fluent from. (*A working directory is the directory where your case and data files reside*)
8. Share your Working directory giving Full Control for permissions. (same instructions as above)
9. Map a Network Drive letter to the shared Working directory. (The mapped drive letter (Z: in my example below) will be used in the FLUENT Launcher)

# Mapping a Network Drive

1. Click Windows Start Button, Computer
2. Click Map network drive
3. Select the Drive Letter
4. Type the name of the computer using this format: \\computer\_name\share\_name (where computer name is the name of your computer and the share\_name is the ANSYS Inc share that you created earlier).
5. Click **Finish** when finished.



# Installing the Message Passing Interface (MPI)

IBM Platform MPI is the default MPI so we recommend that you install this MPI.

Ensure that you have properly installed and configured the necessary Message Passing Interface (MPI). IBM Platform MPI (PCMPI) or INTEL MPI. An MPI is needed for sending data packets back and forth to the nodes. **The MPI must be installed on all computers that you want to submit solve jobs to.**

# Installing IBM Platform MPI or Installing INTEL MPI

1. To install the MPI choose the setup.exe file from the ANSYS Installation.
2. Choose "**Install MPI for ANSYS Parallel Processing**"

Release 16.0

**ANSYS**

Select a language: **English**

**Install ANSYS Products**

**Install ANSYS Documentation Only**

**Install ANSYS EKM Server**

**Install MPI for ANSYS Parallel Processing**

**Install ANSYS License Manager**

Please close all other applications before continuing with the installation.

Getting Started - Installation | Getting Started - Licensing | System Requirements | Installation Help

Exit

ANSYS 16.0 Release

**ANSYS**

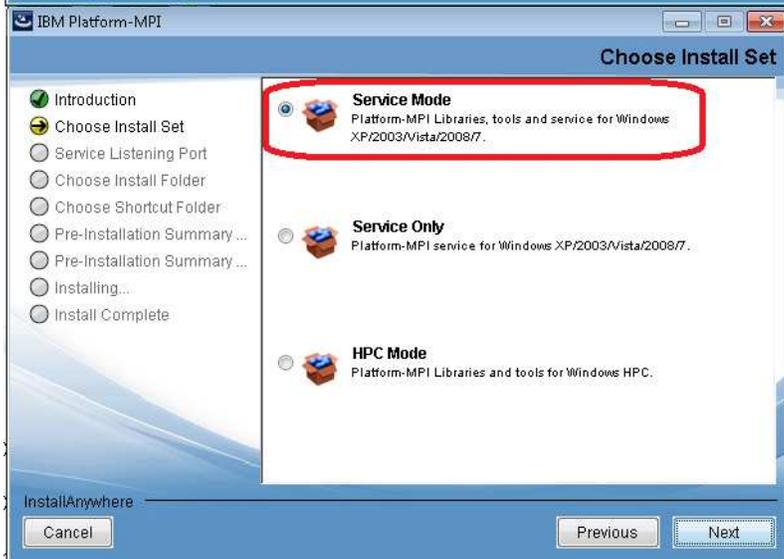
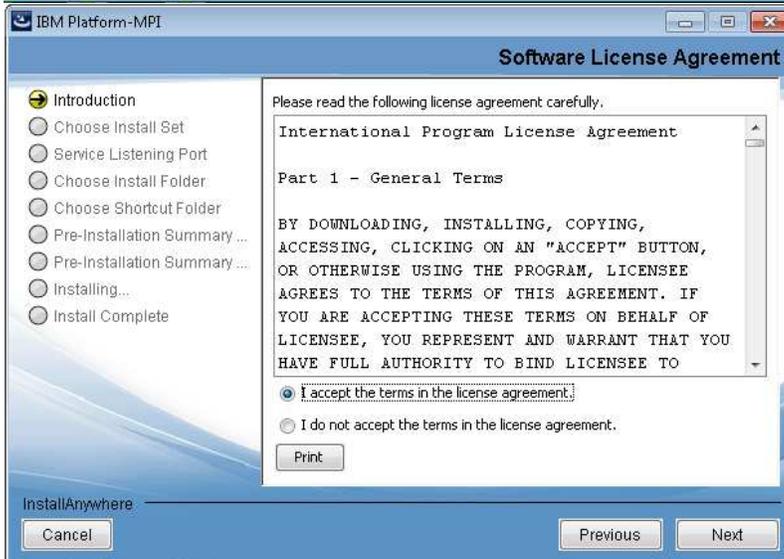
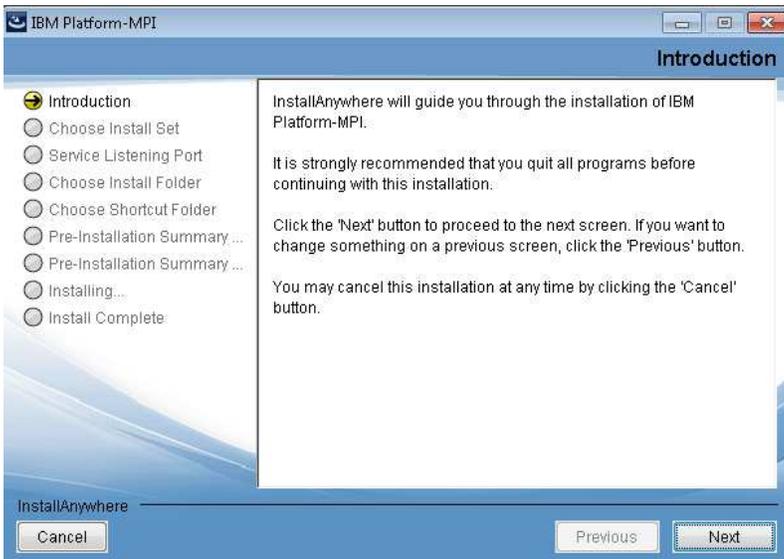
Install MPI only if you are running parallel processing in a distributed mode.

**Install Intel MPI**  
Intel MPI can be used for the FLUENT Parallel, AUTODYN, parallel CPX and Distributed Mechanical APDL products.

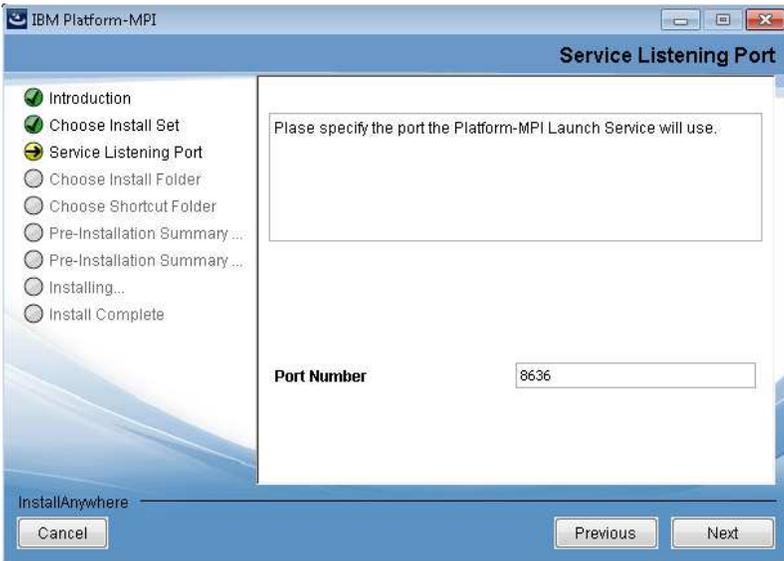
**Install Platform MPI**  
Platform MPI can be used for the parallel Explicit, FLUENT Parallel, parallel CPX, Distributed Implicit, and Distributed Mechanical APDL products.

Back

# Installing IBM Platform MPI



Choose "Service Mode"

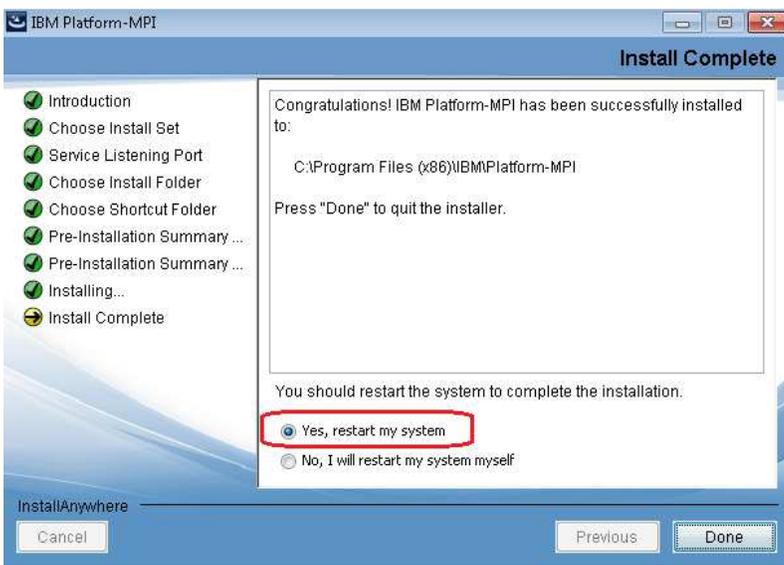


This is the default “**Firewall**” port that IBM Platform MPI will use is 8636. Please add this port to the Firewall Exceptions list if applicable.





Proceed through the installation



Restart your computer.

If you have already installed ANSYS Fluent you can run the MPI installer manually by following the steps below following the steps above once the installer has started.

### Platform MPI (assuming the default installation location)

1. Open up Windows Explorer or File Explorer
2. Browse to **C:\Program Files\ANSYS Inc\v160\commonfiles\MPI\Platform\9.1.2.1\Windows** (do not choose the win64 directory)
3. Run **PlatformMPI-09.1.2.1r.w64.exe** (right-click "Run as Administrator")

### Intel MPI (assuming the default installation location)

1. Open up Windows or File Explorer

2. Browse to **C:\Program Files\ANSYS Inc\v160\commonfiles\MPI\Intel\4.1.3\Windows** (do not choose the winx64 directory)
3. Run **w\_mpi-rt\_p\_4.1.3.045.exe** (right-click “**Run as Administrator**”)

#### **MORE NOTES:**

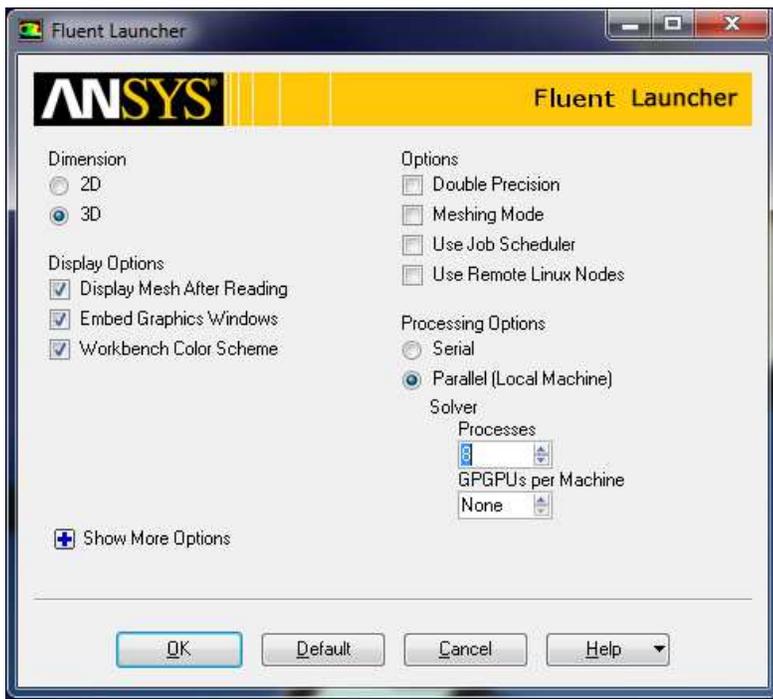
1. If running on one machine you **DO NOT** need to install the MPI
2. If you choose to install Intel MPI follow the same instructions except choose Install Intel MPI.

## **Issue We Have Seen and Best Practices**

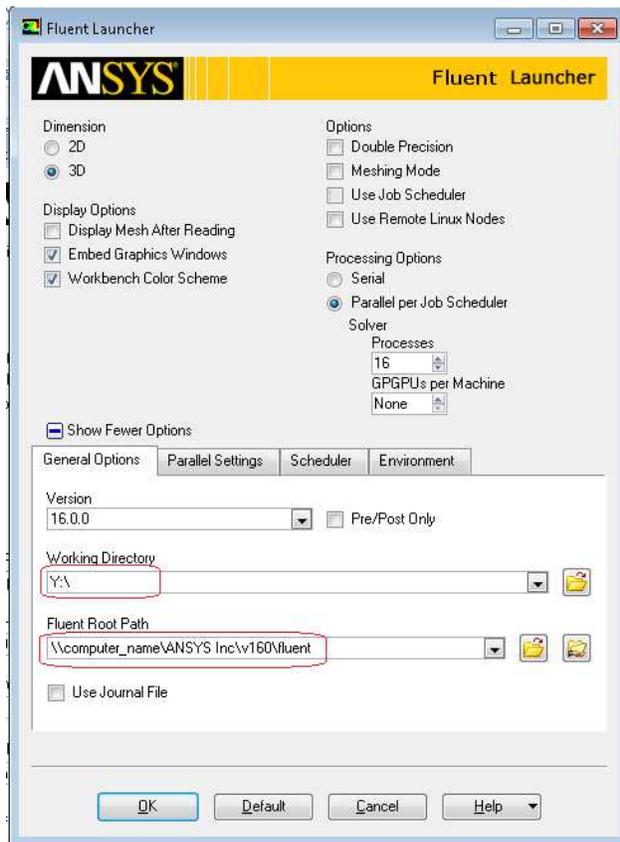
1. Using an installation shared from a Windows XP machine you may see random issues in parallel startup. This is due to Windows setting a very tight limit, 10, for network file connections, which is very easy to exceed. Users are recommended to install ANSYS on a Windows server OS machine and share from this machine.
2. MPIs are sensitive to network issues, different processor levels, memory speeds, operating systems, etc. So the things to check are:
  1. Are all computers on a DOMAIN? MPI's require Active Directory Authentication.
  2. Do all of the Workstations have the same processor type?
  3. The same Memory Speed?
  4. The same Operating System?
  5. If you answered **NO**, to any of these questions, contact ANSYS Support for Best Practices and guidance

## **Launching FLUENT**

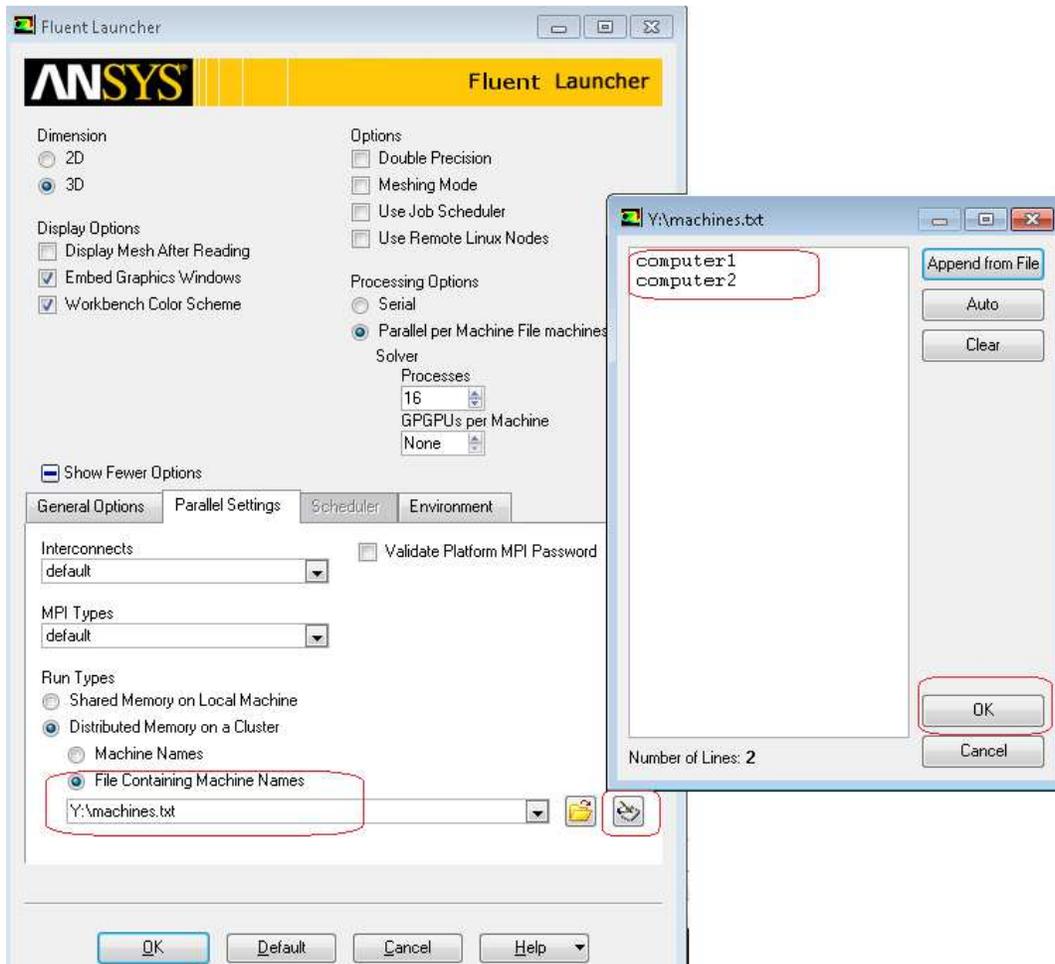
1. Launch FLUENT from **Start Menu, All Programs, ANSYS 16.0, Fluid Dynamics, FLUENT**
2. Select **Parallel** under **Processing Options** and then enter the number of Processes you will be using.
3. Choose "**Show More Options**".



4. On the **General Options** tab use UNC format for the **FLUENT\_Root Path** to point to \\your\_machine\_name\_\share\_name that you set up in the “**Sharing a Directory**” section above.
5. In the **Working Directory** field type in the mapped drive letter you set up in the “**Mapping a Network Drive**” section above.



6. Select the **Parallel Settings** tab.



**NOTE:** Do **NOT** choose "msmpi". This mpi is used exclusively when running Microsoft HPC 2008 OR 2012 Server only. Choose "**Distributed Memory on a Cluster**" followed by "**File Containing Machine Names**"

1. On the **Parallel Settings** tab choose "**File Containing Machine Names**"
2. Click the **create file icon** on the right and type in the name(s) of the computers you want to run distributed on. You only have to type the machine name once. In the example above, you have chosen to run on 16 cores. The machine file would be read and run 8 cores on Computer1 and 8 cores on Computer2.
3. Select **OK** to start Fluent.

## Troubleshooting

### Firewalls

If you have a corporate Firewall turned on then you will not need the local Windows Firewall turned on. If you get a port error with port 8636 in the Fluent window then you will need to disable the Firewalls on all computers or add port 8636 to the Exceptions (This is the IBM Platform MPI default port)

Please see **Solution # 2029819** on the **ANSYS Customer Portal** for more troubleshooting information