

Logiciel ANSYS Fluent sous MacOS

Tutoriel d'installation - ANSYS Student
(DMFAE - S. AUBERT - 20/09/20)





Test de ANSYS Fluent

(temps estimé : 30 minutes)

Test de ANSYS Fluent

Site de référence :

UE Fluides et Energie - Méthodes expérimentales et numériques
pedagogie1.ec-lyon.fr/course/view.php?id=965

Préparation de l'espace de travail :

The image shows two windows side-by-side. The left window is a Windows File Explorer showing the 'Documents' folder. A folder named 'TestFluent' is highlighted in blue. A red box with the number '1' is around the 'Documents' folder in the left sidebar, and another red box with the number '2' is around the 'TestFluent' folder. A red box with the text 'A créer' is overlaid on the File Explorer window. The right window is a web browser showing a page from 'Pédagogie1' with the URL 'https://pedagogie1.ec-lyon.fr/mod/folder/view.php?id=17700'. The page title is 'S5 Fluides et Energie - Méthodes expérimentales et numériques 2020-2021'. A red box with the number '3' is around the breadcrumb trail: 'Tableau de bord / Mes cours / 2020-2021 / Ingénieur de l'Ecole centrale de Lyon / Tronc Commun / Semestre 5 / Enseignement TC / Fluides et Energie / S5 FLE tc 2 2020-2021 / Module 2 : Initiation à la simulation numérique / BE3 Simulation d'écoulement laminaire et turbulent en conduite'. Below the breadcrumb trail, there is a folder view for 'BE3 Simulation d'écoulement laminaire et turbulent en conduite' containing subfolders 'Expansion_Turbulent' and 'profil_keps.prof'. Inside 'Expansion_Turbulent', there are files 'data_expe.zip' and 'expansion_turbulent_keps.cas', which is highlighted with a red box.

Préparation de l'espace de travail (suite) :

The image shows a web browser window displaying a course page for 'S5 Fluides et Energie - Méthodes expérimentales et numériques 2020-2021'. The page lists the course structure, including 'BE3 Simulation d'écoulement laminaire et turbulent en conduite'. A file named 'data_expe.zip' is highlighted in the file list. A dialog box prompts the user to save the file as 'expansion_turbulent_keps.cas'. The file explorer shows the file being saved to the 'TestFluent' directory. Red boxes and arrows highlight the file name and the 'Enregistrer' button.

1

2

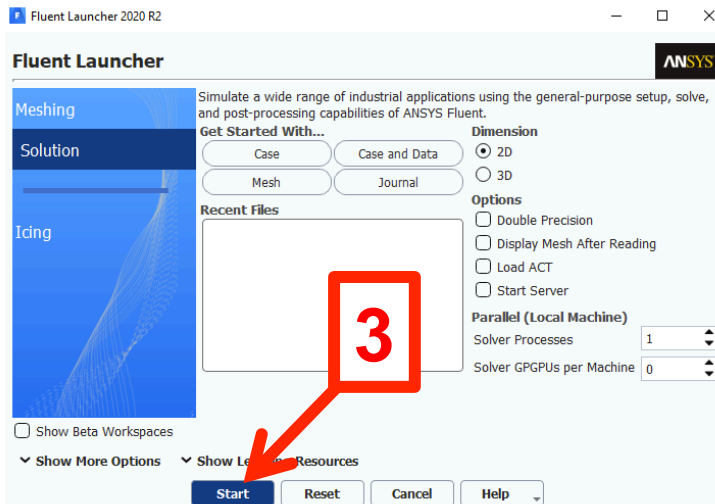
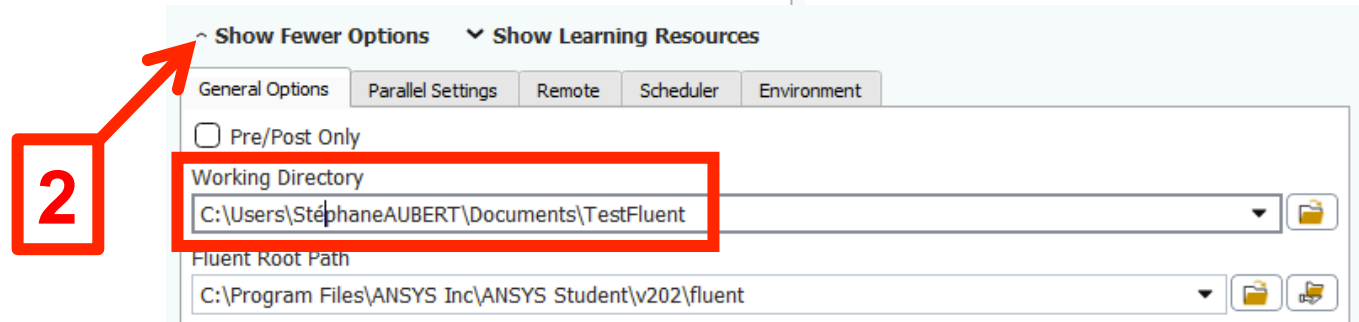
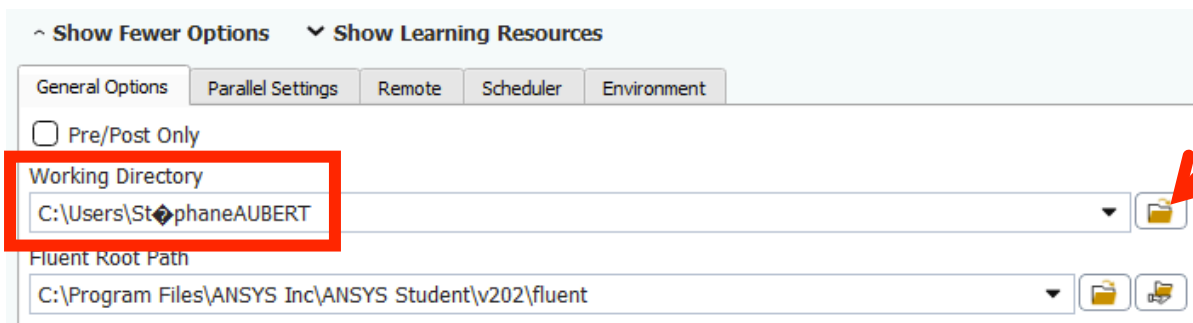
Lancement de Fluent :

The image shows the Windows Start menu on the left and the Fluent Launcher 2020 R2 application window on the right. Red boxes with numbers 1 through 4 and arrows indicate key steps:

- 1**: Points to the Start button in the bottom-left corner of the Windows taskbar.
- 2**: Points to the 'Fluent 2020 R2' application icon in the Start menu list.
- 3**: Points to the 'Dimension' section in the Fluent Launcher window, where '2D' is selected.
- 4**: Points to the 'Show More Options' checkbox at the bottom of the Fluent Launcher window.

The Fluent Launcher window includes sections for 'Meshing', 'Solution', and 'Icing'. The 'Get Started With...' section offers 'Case', 'Case and Data', 'Mesh', and 'Journal' options. The 'Dimension' section has radio buttons for '2D' (selected) and '3D'. The 'Options' section includes checkboxes for 'Double Precision', 'Display Mesh After Reading', 'Load ACT', and 'Start Server'. The 'Parallel (Local Machine)' section has dropdowns for 'Solver Processes' (set to 1) and 'Solver GPGPUs per Machine' (set to 0). At the bottom, there are 'Start', 'Reset', 'Cancel', and 'Help' buttons.

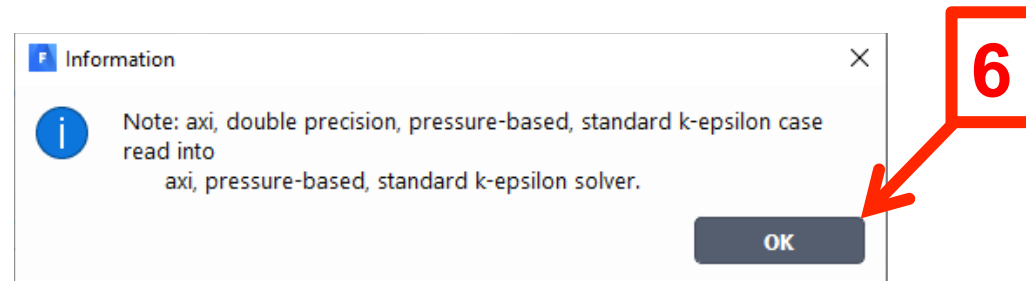
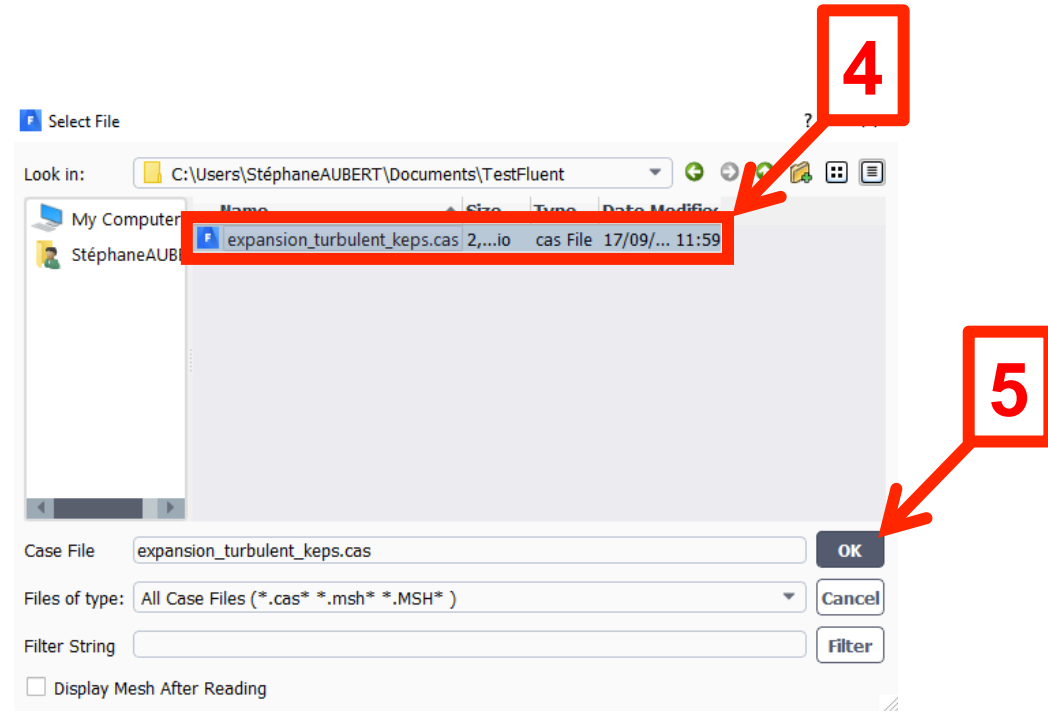
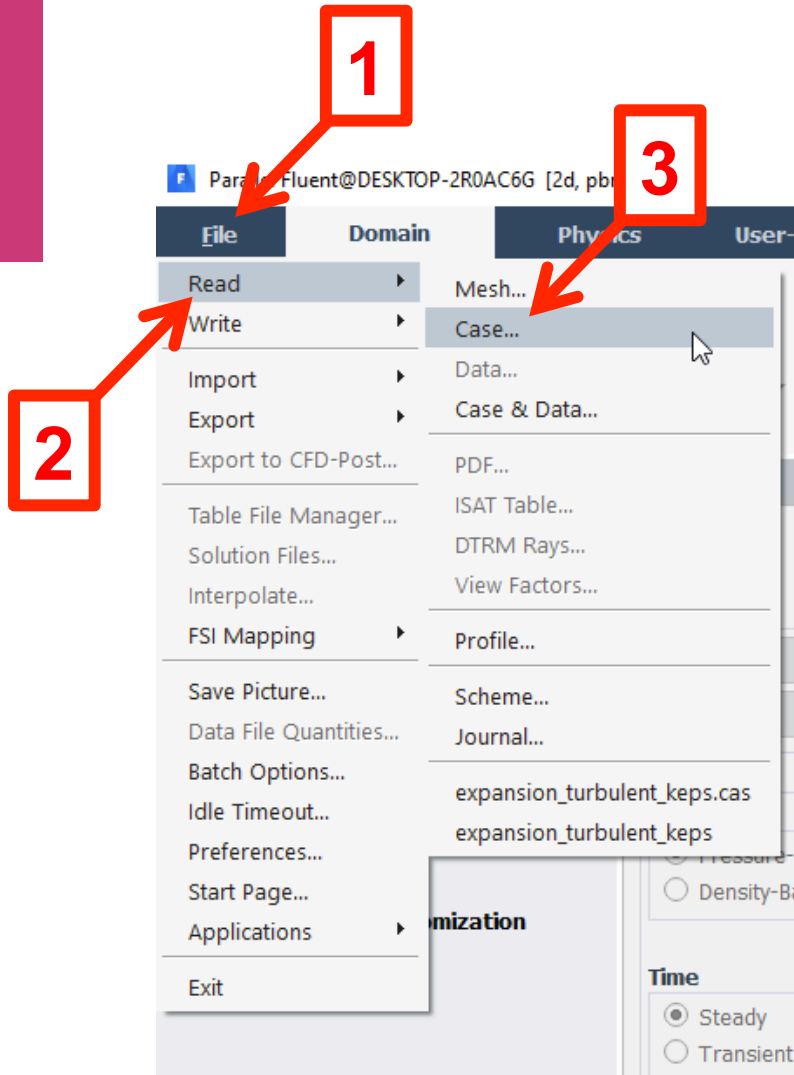
Configuration de l'espace de travail :



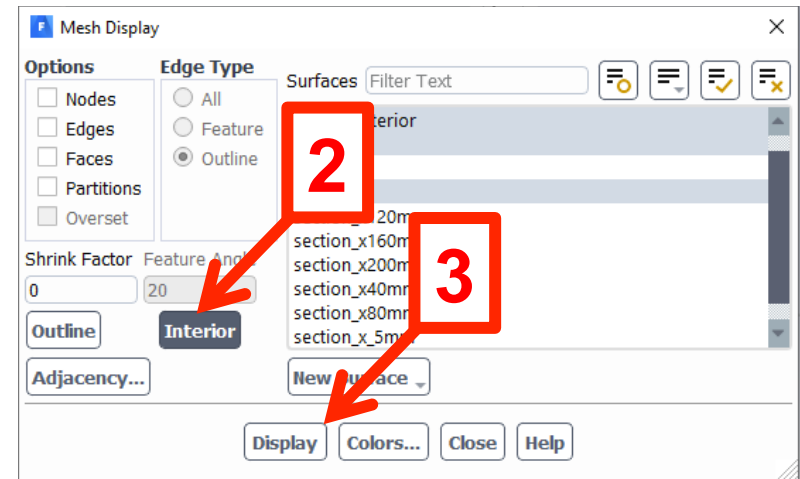
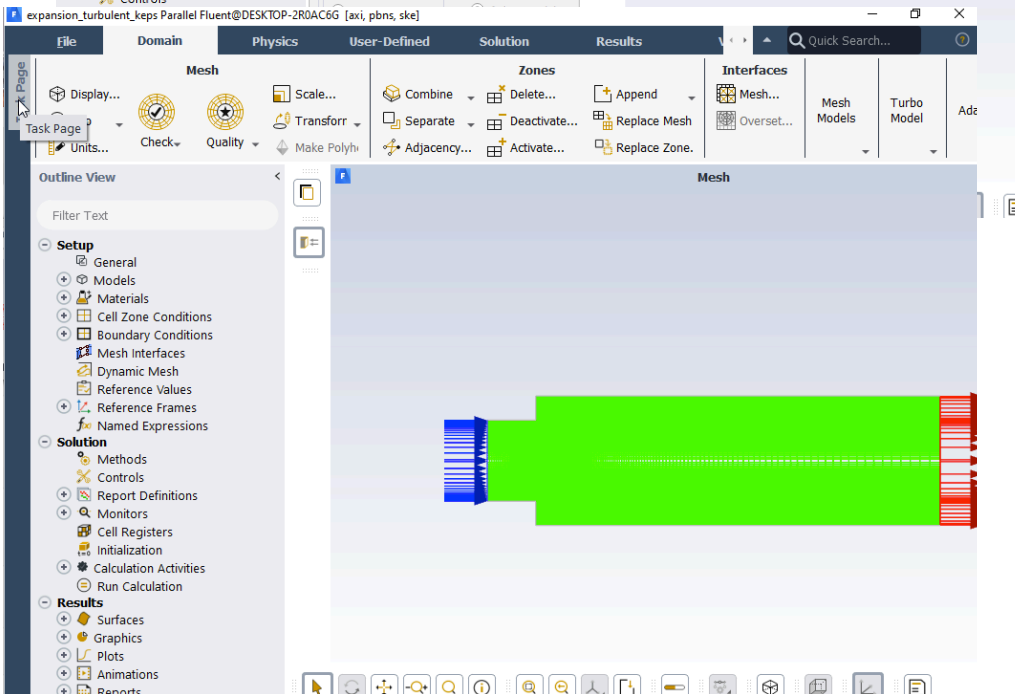
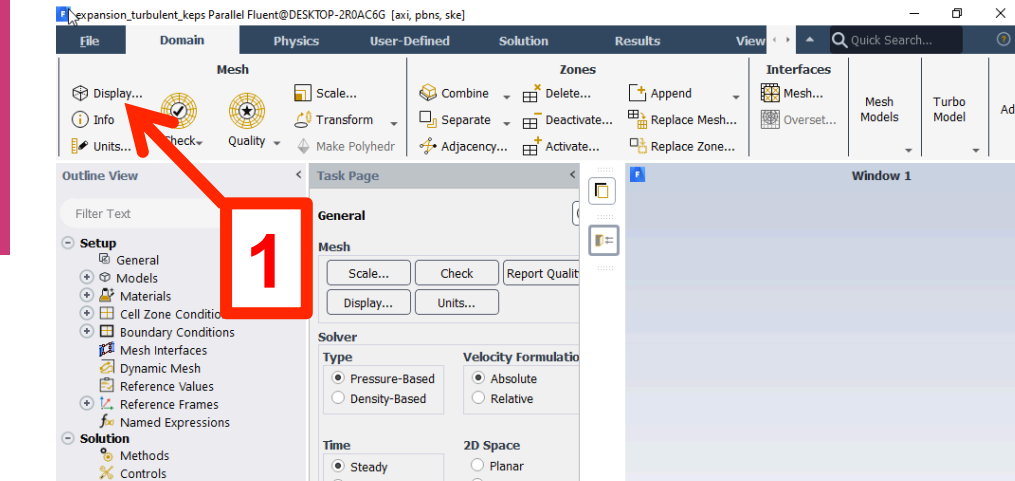
Démarrage de Fluent :

The image displays two screenshots of the ANSYS Fluent software interface. The top screenshot shows a Windows security alert window titled "Alerte de sécurité Windows" (Windows Security Alert) in the foreground. The alert message is in French: "Le Pare-feu Windows Defender a bloqué certaines fonctionnalités de cette application." (Windows Defender Firewall blocked some features of this application). Below the message, it lists details: "Nom : fl2020", "Éditeur : Inconnu", and "Chemin d'accès : C:\program files\ansys\inc\ansys student\lv202\fluent\fluent20.2.0\win64\fd_host\fl2020.exe". Two checkboxes are visible: "Réseaux privés, tels qu'un réseau domestique ou un réseau d'entreprise" (checked) and "Réseaux publics, tels qu'un aéroport ou un cybercafé (non recommandé car ces réseaux sont rarement sécurisés)" (unchecked). A red box labeled "1" highlights the alert title bar, and another red box labeled "2" highlights the "Autoriser l'accès" (Allow access) button. The background shows the Fluent software interface with the "Task Page" open to the "General" section. The bottom screenshot shows the same software interface with the "Task Page" open to the "Mesh" section, displaying various meshing options like "Scale...", "Check", and "Report Quality".

Lecture du cas test :



Affichage du maillage et des conditions limites :



Résolution numérique :

expansion_turbulent_keps Parallel Fluent@DESKTOP-2R0AC6G [axi, pbns, ske]

File Domain Physics User-Defined Solution Results View Quick Search...

Mesh Scale... Transform... Make Polyhedr... Combine... Delete... Append... Mesh... Mesh Models Turbo Model Ada... Deactivate... Replace Mesh... Activate... Replace Zone...

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation

Results

- Parameters & Customization

Run Calculation

Check Case... Update Dyna...

Pseudo Transient Settings

Fluid Time Scale

Time Step Method: Automatic Time Scale Factor: 1

Length Scale Method: Conservative Verbosity: 0

Parameters

Number of Iterations: 10

Reporting Interval: 1

Profile Update Interval: 1

Solution Processing

Statistics

Data Sampling for Steady Statistics

Solution Advancement

Calculate

Question

The solution must be initialized before the calculation can be started. Do you want to proceed after initializing the case now?

Yes No

expansion_turbulent_keps Parallel Fluent@DESKTOP-2R0AC6G [axi, pbns, ske]

File Domain Physics User-Defined Solution Results View Quick Search...

Mesh Scale... Transform... Make Polyhedr... Combine... Delete... Append... Mesh... Mesh Models Turbo Model Ada... Deactivate... Replace Mesh... Activate... Replace Zone...

Outline View Task Page

Filter Text

Setup

- General
- Models
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Mesh Interfaces
- Dynamic Mesh
- Reference Values
- Reference Frames
- Named Expressions

Solution

- Methods
- Controls
- Report Definitions
- Monitors
- Cell Registers
- Initialization
- Calculation Activities
- Run Calculation

Results

- Parameters & Customization

Run Calculation

Check Case... Update Dyna...

Pseudo Transient Settings

Fluid Time Scale

Time Step Method: Automatic Time Scale Factor: 1

Length Scale Method: Conservative Verbosity: 0

Parameters

Number of Iterations: 10

Reporting Interval: 1

Profile Update Interval: 1

Solution Processing

Statistics

Data Sampling for Steady Statistics

Solution Advancement

Calculate

Scaled Residuals

report-def-0

iteration

Information

Calculation complete.

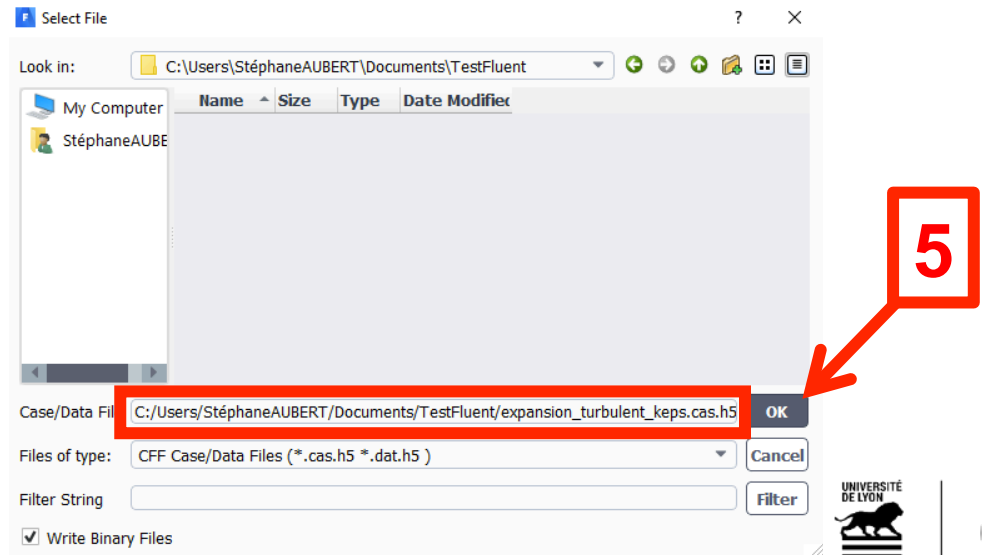
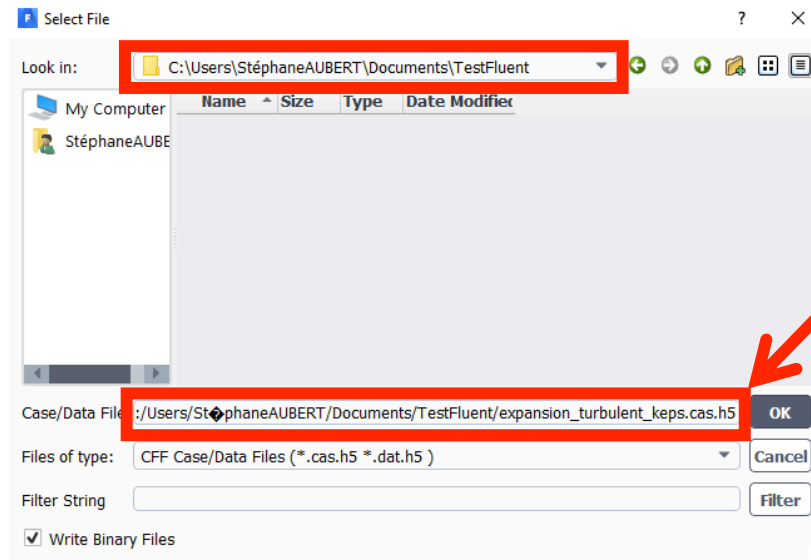
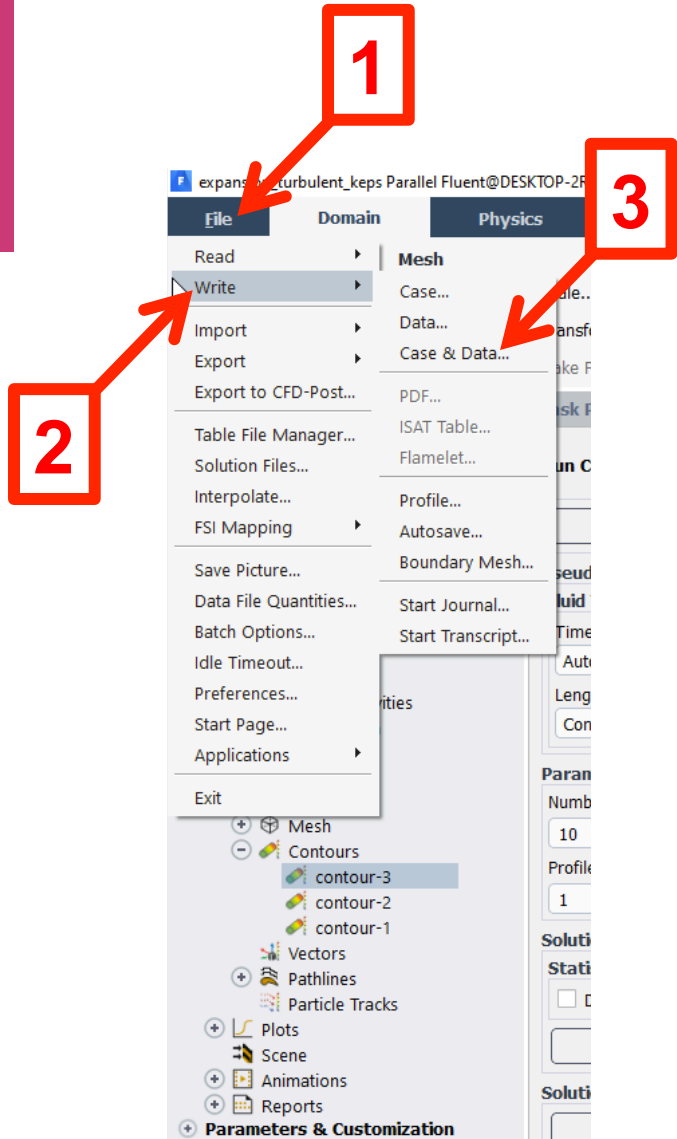
OK

A vérifier

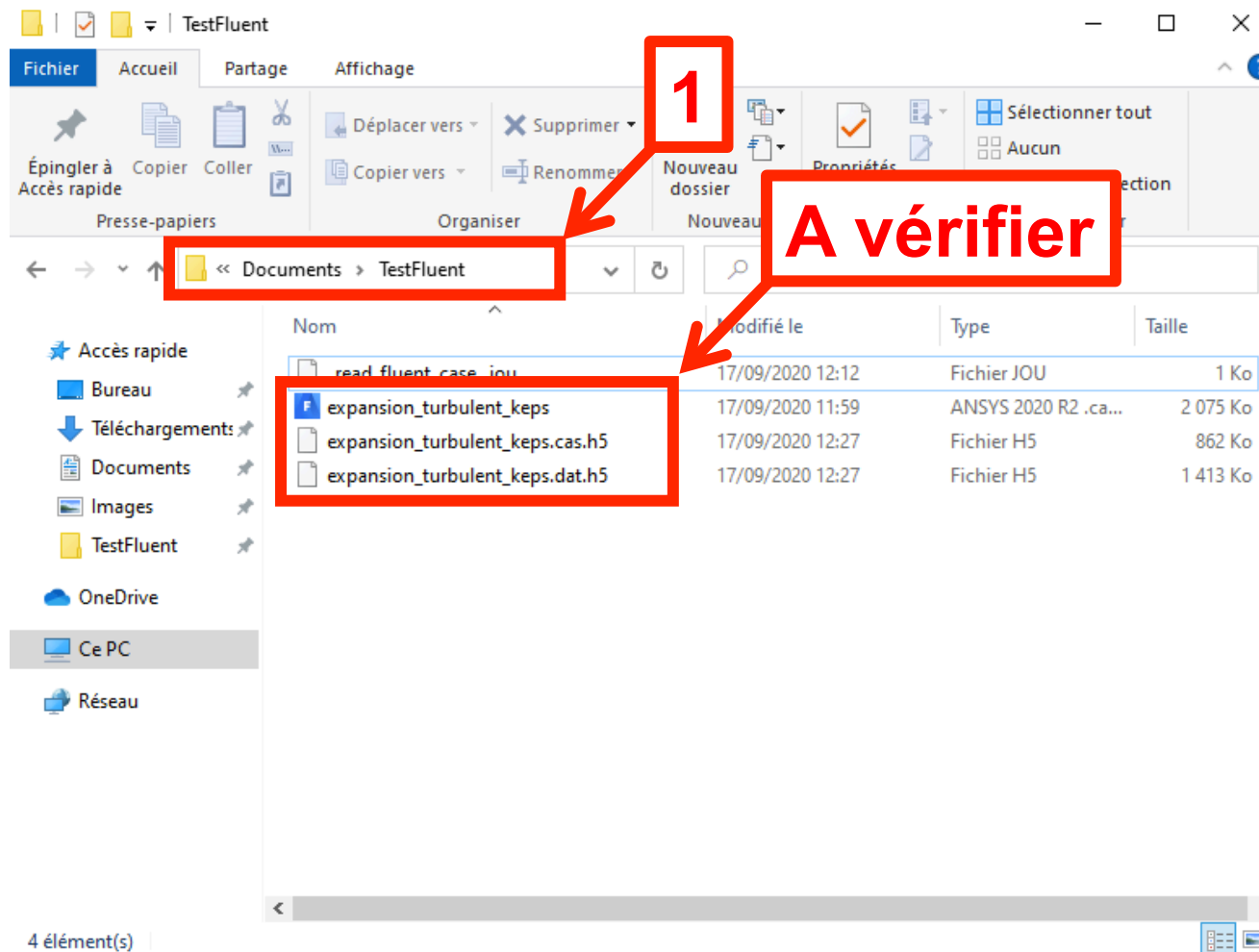
Visualisation de la solution :

The image displays the ANSYS Fluent software interface during a simulation. The 'Contours' dialog box is open, showing the 'Contours of' dropdown set to 'Turbulent Kinetic Energy (k)'. The 'Save/Display' button is highlighted with a red arrow and box labeled '3'. In the 'Results' window, the 'Contours' folder is expanded, and 'contour-3' is selected, indicated by a red arrow and box labeled '1'. A red box labeled '2' points to the 'Contours of' dropdown in the dialog box. The 'Results' window shows a contour plot of 'Turbulent Kinetic Energy (k)' with a color scale ranging from 1.65e-07 to 3.33e-01. The plot shows two distinct regions of high energy (red/yellow) in a blue fluid domain. The 'Iteration' plot shows the convergence of the solution over 10 iterations.

Enregistrement de la solution :



Contrôle de l'enregistrement :





ÉCOLE
CENTRALE LYON

36, avenue Guy de Collongue 69130 Écully - France
+33 (0)4 72 18 60 00

www.ec-lyon.fr