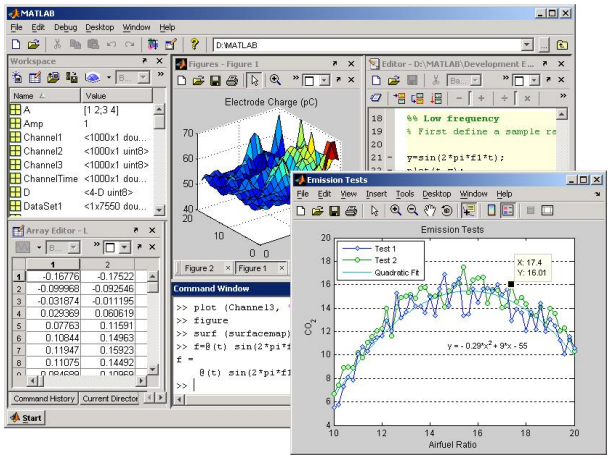
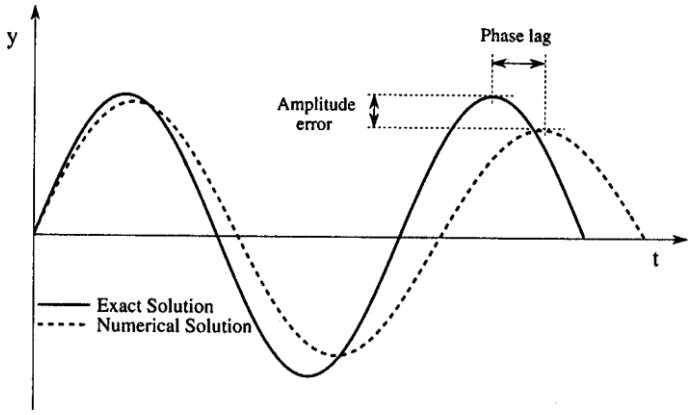
UE 703 Méthodes Numériques pour la Mécanique

POLYCOPE DE TP – M1 ENERGIE





Olivier BOTELLA

LEMTA – Université de Lorraine - CNRS
2 avenue de la Forêt de Haye, TSA 60604
54518 Vandoeuvre (France)

Email : Olivier.Botella@univ-lorraine.fr

Lectures web page : https://arche.univ-lorraine.fr/course/view.php?id=6745

Analyse Numérique - Master MEPP

Enoncés des TP - Introduction à MATLAB pour le calcul numérique

olivier.botella@univ-lorraine.fr

0.1 Organisation des TP

Les TP d'analyse numérique déroulent en 8 séances de 3h de la manière suivante :

- 5 séances d'analyse numérique avec MATLAB ,
- 3 TP d'initation à FLUENT, le logiciel commercial de simulation en mécanique des fluides et thermique.

Si vous n'avez aucun connaissance en MATLAB , il est recommandé de réaliser les TP d'introduction dont l'énoncé se trouve dans ce poly. Pour réaliser les TP d'analyse numérique, vous devez télécharger sur ARCHE (http://arche.univ-lorraine.fr) l'archive '`tpmatlab_m1mepp.zip`' du cours *UE104* et l'extraire sur votre PC. Cette archive contient la documentation et les programmes MATLAB à utiliser et compléter lors des séances.

0.2 Notation des travaux pratiques

Les TP seront notées d'une part par une de votre travail et assiduité durant les séances, et sur la base de compte-rendus individuels que vous devez rendre à la fin du semestre.

lieu à 2 notes : une pour l'EC *'Introduction à MATLAB '*, et une note de TP pour l'EC *'Analyse numérique'*.

- Les compte-rendu devront répondre aux questions de l'énoncé, et comporter 6 pages au maximum. Les réponses doivent être rédigées de façon précise et succinte (ne recopiez pas l'énoncé !), et seront illustrées par les graphiques que vous avez réalisé avec MATLAB . Dans certains cas, il vous sera demandé de rendre le listing de votre programme MATLAB .
- Les compte-rendu doivent être rédigés avec le traitement de texte WORD ou convertis en fichiers PDF. Ne rendez pas de fichiers MATLAB . N'oubliez pas de mettre votre nom, votre parcours licence et l'intitulé du TP sur les compte-rendu.

Analyse Numérique - Master MEPP

Introduction à MATLAB pour le calcul numérique

olivier.botella@univ-lorraine.fr

1 Prise en main de MATLAB

1.1 Introduction

MATLAB (abbréviation de *MATrix LABoratory*) est un système interactif, basé sur le calcul matriciel, qui permet de réaliser des calculs scientifiques sans nécessiter d'écrire de longs programmes informatiques. Le but de ces séances de TP est de vous introduire aux fonctionnalités de base de MATLAB dans l'optique de son utilisation en analyse numérique. Néanmoins, MATLAB vous sera utile non seulement en calcul scientifique, mais aussi pour l'exploitation de données expérimentales.

1.2 Premiers pas dans MATLAB

Pour réaliser ces TP, vous devez télécharger sur ARCHE (http://arche.univ-lorraine.fr) l'archive '`tpmatlab_m1mepp.zip`' du cours *UE 104* et l'extraire sur votre PC. Cette archive contient la documentation et les programmes MATLAB à utiliser et compléter lors des séances.

Sur votre PC, allez dans le répertoire de travail *'TP0 INITIATION MATLAB'* qui contient les programmes MATLAB (fichiers **.m*) pour les séances d'initiation. Le fichier *'Tp0_matlab.m'* contient les commandes (ou *instructions*) MATLAB utiles au premier TP. Ouvrez ce fichier par double-clic, et l'interface graphique représentée sur la figure 1 apparaîtra à l'écran.

L'interface graphique de MATLAB se décompose en :

- **Command Window** : pour entrer les commandes MATLAB (après le prompt >>) et voir les résultats s'afficher.
- **MATLAB Editor** : pour éditer les fichiers **.m* contenant les commandes et fonctions MATLAB .
- **Current Folder** : affiche l'ensemble des fichiers **.m* contenus dans le répertoire de travail.
- **Workspace** : affiche les données, matrices, *etc* ... que vous créez durant la séance de travail.
- **Command History** : historique des commandes de la séance de travail.

1.3 Exécution d'un programme MATLAB

Il y a 3 manières équivalentes d'exécuter le programme *'Tp0_matlab.m'* :

- Taper `>> Tp0_matlab` dans la **Command Window**,
- Cliquer sur le bouton  dans la barre du menu EDITOR de l'interface graphique,
- Placer la souris dans le **MATLAB Editor** du programme *'Tp0_matlab.m'* et taper F5 au clavier.

Le résultat du programme s'affiche dans la **Command Window**. Il est à noter que le programme *'Tp0_matlab.m'* utilise une fonction définie dans le fichier *'fonction_ex0.m'* qui doit nécessairement se trouver dans le répertoire de travail, sinon MATLAB ne trouvera pas la fonction et affichera un message d'erreur.

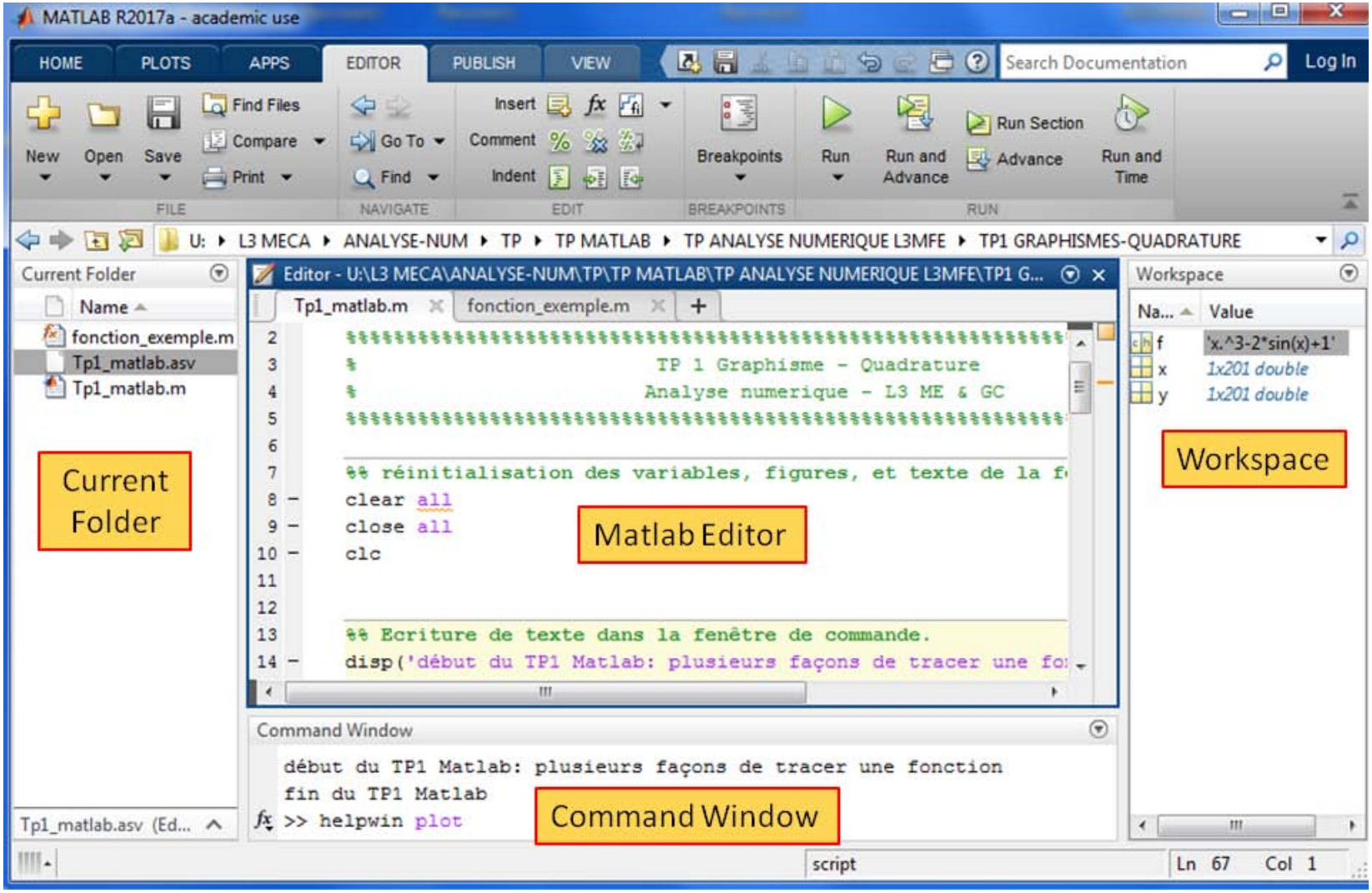


Figure 1: Interface graphique de MATLAB .

2 TP d'initiation à MATLAB

2.1 Introduction

Vous devez effectuer en 3 séances les exercices donnés ci-dessous. Pour cela, vous devez lire en premier le polycopié *'Introduction à MATLAB '* et exécuter tous les exemples qu'il contient. Il sera souvent nécessaire de rechercher des informations complémentaires au poly dans l'aide en ligne de MATLAB :

```
>>helpwin commande
```

2.2 Enoncés des exercices d'initiation

Exercice 1. ***(Algèbre linéaire et calcul matriciel)***

On considère la matrice rectangulaire $\mathcal{M}$ définie par :

$$\mathcal{M} = \begin{bmatrix} 4 & 8 & 12 \\ 3 & 8 & 13 \\ 2 & 9 & 18 \\ -1 & 0 & 1 \end{bmatrix}$$

1) Créez la matrice $\mathcal{M}$ dans MATLAB . Donnez le nombre de lignes et de colonnes (commande size).

2) Extrayez de $\mathcal{M}$ la matrice carrée :

$$\mathcal{A} = \begin{bmatrix} 4 & 8 & 12 \\ 3 & 8 & 13 \\ 2 & 9 & 18 \end{bmatrix}$$

3) On considère le système linéaire $\mathcal{A}X = F$, tel que le second membre F est défini par :

$$F = \begin{pmatrix} 4 \\ 5 \\ 11 \end{pmatrix}$$

Montrez que le système possède une solution unique en calculant son déterminant et résolvez le système linéaire.

4) Vérifiez que la solution donnée par MATLAB est exacte en calculant le résidu $R = F - \mathcal{A}X$.

Exercice 2. ***(Graphisme sous MATLAB)***

Cet exercice a pour objet d'initier aux multiples possibilités graphiques de MATLAB . Utilisez l'aide en ligne (helpwin plot) pour le détail des options disponibles.

1) Sur une même fenêtre graphique (commande subplot) tracez les trois graphes représentant dans $[-\pi, \pi] \times [-1, 1]$ la fonction $x \mapsto sin(x)$ et ses développements de Taylor autour de $x = 0$, *i.e.* $x \mapsto x$ et $x \mapsto x - x^3/3$. Vous devrez respecter la mise en forme de la figure 2, c'est-à-dire que chaque graphe ait une couleur et un trait différent, et que les figures soient correctement légendées.

2) Utilisez la barre des taches de la fenêtre graphique MATLAB pour exporter les graphes sous format *jpeg* ou *png*, ainsi que pour faire un zoom autour du point $x = 0$.

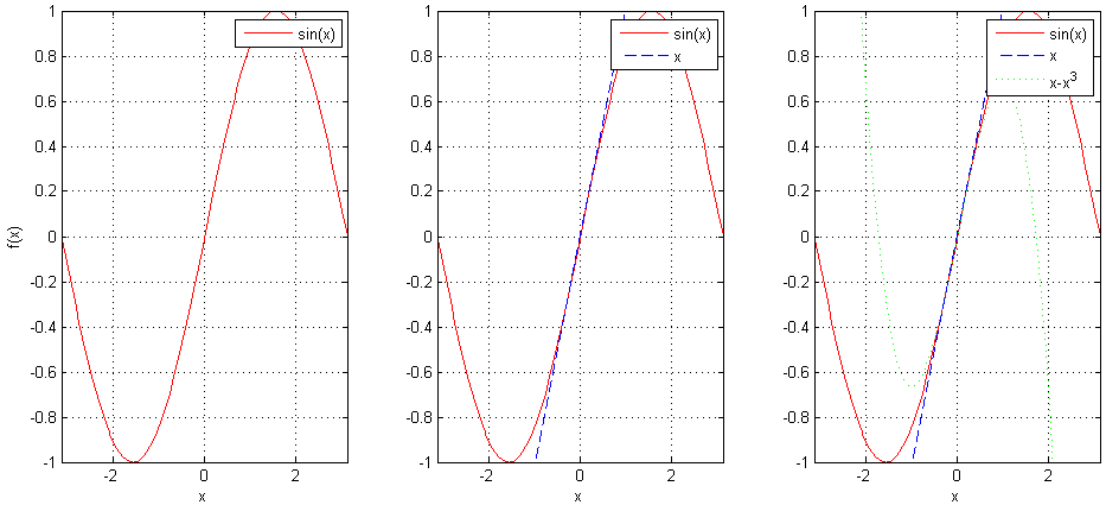


Figure 2: Graphisme obtenu sous Matlab

.

Exercice 3. ***(Fonctions MATLAB et boucles de contrôle)***

1) A l'aide de la boucle for, Ecrivez la fonction MATLAB F=fact(n) qui calcule la factorielle de l'entier n (Faites un test conditionnel (boucle if) qui calcule $n!$ seulement si $n \geq 0$). *Application :* calculez la factorielle de -1, 0 et 3.

2) Ecrivez la fonction R=residumat(A,X,F) qui calcule le résidu du système linéaire $\mathcal{A}X = F$ de taille $n \times m$, soit :

$$\text{Pour } i = 1, \dots n, \qquad R_i = F_i - \sum_{j=1}^{m} A_{i,j} X_j$$

Testez-la sur l'exemple de l'exercice 1.

3) Ecrivez la fonction MATLAB `[r1,r2]=racines_poly(a,b,c)` qui calcule les racines *réelles* du pôlynome du second degré $ax^2 + bx + c$, en faisant un test suivant la valeur du discriminant Δ. Appliquez-le au calcul des racines de $x^2 + x - 2$.

Exercice 4. ***(Intégration numérique par méthode du trapèze)***

1) Ecrivez la fonction MATLAB trapeze(a,b,f,n) qui calcule l'intégrale $\int_a^b f(x)\mathrm{d}x$ par la formule du trapèze composite avec n sous intervalles de taille uniforme[1].

2) Validez la fonction que vous avez écrite en vérifiant qu'elle donne l'intégrale exacte d'une fonction linéaire (polynôme de degré 1).

[1] On rappelle que, d'après le cours, cette formule s'écrit :

$$I_h = \frac{h}{2} \sum_{k=0}^{n-1} \left[f(x_k) + f(x_{k+1}) \right],$$

avec $h = (b-a)/n$ et $x_k = a + kh$.

3) Vous allez maintenant utiliser votre fonction trapeze(a,b,f,n) pour calculer l'intégrale

$$I = \int_1^{\pi} \frac{\sin x}{2x^3}\,\mathrm{d}x \simeq 0.198557\ldots \tag{1}$$

qui a été vue en TD d'analyse numérique. Il est possible de calculer la valeur exacte de cette intégrale à l'aide de la *toolbox* de calcul formel symbolic de MATLAB , en tapant ces lignes :

```
>> syms x
>> I=double(int(f_exo_trapeze(x),1,pi))
```

a) Pour un nombre de sous-intervalles n suffisament grand, calculez le facteur de réduction de la méthode du trapèze et vérifiez qu'on retrouve le résultat théorique.

b) Pour $n \in [50, 200]$, représentez en échelle semi-logarithmique (commande semilogy) l'erreur numérique $E_n = |I_n - I|$ de la méthode du trapèze et montrez qu'elle est au second-ordre en traçant sur le même graphe la fonction $n \mapsto 1/n^2$.

MATLAB Quick Reference

Operators

Matrix Operations		Array or Element by Element	
+	Addition		
-	Subtraction		
*	Matrix Multiplication	.*	Element by Element Multiplication
/	Right Matrix Division $b/A = bA^{-1}$	./	Element by Element Right Division
\	Left Matrix Division $A\backslash b = A^{-1}b$	.\	Element by Element Left Division $A.\backslash B = B./A$
^	Raise Matrix to a power	.^	Raise each Element to a power
'	Transpose matrix (conjugate if complex) A'	.'	Transpose matrix (no complex conjugation) $A.'$

Relational Operators		Logical Operators	
<	Less Than	&	And
<=	Less Than or Equal	|	Or
>	Greater Than	~	NOT
>=	Greater Than or Equal		
==	Equal		
~=	Not Equal		

Special Number Symbols

pi	π
inf	∞
NaN	Not a number ie 0/0

i	$\sqrt{(-1)}$
j	$\sqrt{(-1)}$

Entering Matrices

x = [1 2 3; 4 5 6; 7 8 9]	Comma or space between elements. Semicolon or return for new row. Enclosed in square brackets.
x = [1,2,3 4,5,6 7,8,9]	
a = [exp(0) sqrt(4) 1+2]	Each element can be an expression.
C = [A B]	A and B are matrices with the same number of rows.
C = [A ; B]	A and B are matrices with the same number of columns.

Indexing

Vector Indexing	
variable(V)	V is a vector of indexes
Examples	
A(3)	The third element in vector A.
A([3 8 11])	The third, eighth and eleventh elements of A.
A(3:11)	The third to eleventh elements of A.
A(11:end)	All elements from the eleventh to the last element

Matrix Indexing	
variable(R,C)	R is a vector of rows and C a vector of Columns
Examples	
M(2,3)	The element in the second row and third column of M.
M(2, [3 8 11])	Second row, third, eighth and eleventh column.
M(4,3:11)	Forth row, third to eleventh column.
M(11:end,2)	Eleventh to the last row, second column.
M(5,:)	Fifth row, all columns.
M(11:end,:)	Eleventh to the last row, all columns.

Output Display Format

format short	4 decimal places	format long	15 decimal places
format	Same as above		
format shortE	Scientific notation, 4 decimal places	format longE	Scientific notation, 15 decimal places
format shortEng	Engineering notation, 4 decimal places	format longEng	Engineering notation, 15 decimal places

Generating Vectors

<start> : <end>	**x = 1 : 5** generates x = [1 2 3 4 5]
<start> : <separation> : <end>	**y = 0:5:20** generates y = [0 5 10 15 20]
linspace(start,end,n) n = number of elements	**linspace(0,10,5)** generates [0 2.5 5 7.5 10]
logspace(d1,d2,n) n elements logarithmically spaced between 10^{d1} and 10^{d2}.	**logspace**(-1,2,4) generates [0.1 1 10 100]

Utility Matrices

zeros(n)	n by n matrix where each element is zero.
zeros(m,n)	m by n matrix where each element is zero.
zeros(a,b,c)	3 dimensional array, a by b by c.
ones(m,n)	m by n matrix where each element is one.
rand(m,n)	m by n matrix of random numbers.
eye(n)	n by n identity matrix.

Variable Control

who	List all variables in memory.
whos	Same as above but with more information.
clear	Remove all variables from memory.
clear <*variable*>	Remove specified variables from memory.

File Control Commands

dir	List contents of current directory.
ls	List contents of current directory.
what	List the Matlab files in the current directory.
cd <*directory*>	Change the current directory.
type <*filename*>	Display the contents of a text or .m file.
delete <*filename*>	Delete a file.
diary <*filename*>	Record all commands and results to a file.
diary off	Stop above.

Help

help	Display help topics.
help <function>	Help on a particular function.
lookfor <word>	Look for word in function descriptions.
doc <function>	Full documentation on function.

In Built Functions

Hit ***fx*** icon next to prompt for function browser. Only selected functions shown here.

abs(x)	The absolute value. Modulus	**round(x)**	Round to the nearest integer.
sqrt(x)	The square root.	**ceil(x)**	Round up.
exp(x)	The exponential base e. e^x	**floor(x)**	Round down.
log(x)	The natural logarithm. $\log_e(x)$	**fix(x)**	Round towards zero.
log10(x)	The log base 10. $\log_{10}(x)$	**rem(x,b)**	Remainder of x divided by b.

Trigonometry

sin(x)	Sine. x in radians.		
sind(x)	Sine. x in degrees.		
asin(x)	The arcsine. The inverse of sin(x). Radians		
asind(x)	The arcsine. Degrees		
sinh(x)	Hyperbolic Sine.		
asinh(x)	The inverse Hyperbolic Sine.		
The above variations are also available for the following functions.			
cos(x)	Cosine		
tan(x)	Tangent	**cot(x)**	Cotangent
sec(x)	Secant	**csc(x)**	Cosecant

Complex Numbers

real(z)	The real part of z.	**imag(z)**	The imaginary part of z.
abs(z)	The modulus of z.	**angle(z)**	The phase angle of z.
conj(z)	The complex conjugate of z.		

Matrix

det(A)	Determinant	**sqrtm(A)**	The matrix square root.
norm(A)	Norm	**expm(A)**	The matrix exponential base e.
inv(A)	Inverse	**logm(A)**	The matrix natural logarithm.
[v,d]=eig(A)	d = Eigenvalues, v = Eigenvectors		

Statistics

max(x)	Maximum	**median(x)**	Median	**var(x)**	Variance
min(x)	Minimum	**mean(x)**	Average	**std(x)**	Standard Deviation

Polynomials

p = [1 2 3 4 5]; can represent the polynomial $x^4+2x^3+3x^2+4x+5$	
y = polyval(p,x)	Evaluate polynomial for each value in x.
roots(p)	Roots of polynomial.
p = poly(<roots>)	Polynomial with given roots.
p = polyfit(x,y,n)	Best fit of **x,y** data points to n^{th} order polynomial.

Saving, Exporting and Importing Data

save <*filename***>**	Save all variable to the file *filename*.mat
load <*filename***>**	Load in variables from the file *filename*.mat
save <*filename***> <***variable***>**	Save only the variable *variable* to the file *filename*.mat
save <*filename***> <***variable***> -ascii**	Save *variable* to the text file *filename*
load <*filename***>.<***ext***>**	Load from the text file, to a variable called *filename*

Misc

x = input('What is x ? ')	Ask the user for a number.
[x,y] = ginput(1)	Graph coordinates of a clicked on point.
pause	Wait for the user to press a key.
pause(5)	Wait for 5 seconds
! command	Execute an operating system command
display(a)	Suppose a = 5. This would print "a = 5";
disp(a)	Same as above, but with out the "a = ".

fprintf Formatting

%f	Fixed point	**%5f**	5 characters wide	**\n**	New line
%e	Exponential Notation	**%5.2f**	2 decimal places (dp)	**\t**	Horizontal tab
%g	auto chooses %f or %e	**%-5.2f**	Left justify	****	Back slash
%s	String of Characters	**%+5.2f**	Print sign (+ or -)	**%%**	Percent character
%d	Integer (Whole numbers only)				

Examples	What is printed	Comment
fprintf(' x = %f ',pi)	**x = 3.141593**	No new line after
fprintf(' x = %f \n',pi)	**x = 3.141593**	Moves to a new line after
fprintf(' L%6.2fR \n',pi)	**L 3.14R**	Number is 6 chars wide with 2 dp
fprintf('%s L%-6.2fR','Hi',pi)	**Hi L3.14 R**	The string **'Hi'**, then left justify number
S = sprintf(' x = %f ',pi);	NOTHING	**S** is a string containing **' x = 3.141593'**

Graph Commands

plot(y)	Plot y against index number.
plot(x,y)	Plot y against x
plot(x1,y1,x2,y2)	Plot y1 against x1 and y2 against x2.
plot(x,y,'r+')	Plot y against x using red plus signs.
plot(x1,y1,'r+',x2,y2,'go')	Red plus signs for x1 and y1, Green circles for x2 y2.

Symbol	Line Type or Mark	Symbol	Colour
.	Point	**r**	Red
o	Circle	**g**	Green
x	X mark	**b**	Blue
+	Plus sign	**y**	Yellow
*****	Stars	**m**	Magenta
-	Solid line	**c**	Cyan
:	Dotted line	**w**	White
-.	Dash dot line	**k**	Black
--	Dash Line		

Other Types of Plot	
fill(x,y,'r')	Red filled graph
bar(x,y)	Bar graph
loglog(x,y)	x & y log scale
semilogx(x,y)	x log, y linear
semilogy(x,y)	x linear, y log
polar(theta,r)	Polar plot
surf(x,y,z)	3D surface
mesh(x,y,z)	3D mesh
plot3(x,y,z)	3D line plot

Other Graphics Commands	
title('Title')	Graph title.
xlabel('X axis')	Label the x-axis.
ylabel('Y axis') zlabel('Z axis')	Label y and z axis.
text(x,y,'My Text')	Place text at coordinates x,y.
grid	Place a grid on the graph.
hold on	Add any new plot to the current graph.
hold off	Replace current plot with any new plot.
subplot(r,c,n)	Split figure into r rows by c columns of subplots.
axis([minX maxX minY maxY])	Set the limits of the graph in X and Y. (Note 4 element vector)
h = figure	New graphics window.
figure(h)	Change to plotting in figure h.
delete(h)	Delete figure h.
clf	Clear current figure.
drawnow	Force the graph to update now.

subplot(2,3,1)	subplot(2,3,2)	subplot(2,3,3)
subplot(2,3,4)	subplot(2,3,5)	subplot(2,3,6)

Programming

Enumerated Loops (for)

The general form of a **for** loop is :-

```
for <variable> = <vector>
   <statement>
   <statement>
      etc
end
```

Examples

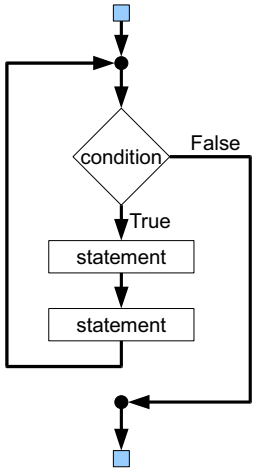
```
A = rand(1,5);
for a = A
   disp(a)
end
```

```
for k = [ 1 7 3 pi i]
   disp(k)
end
```

Precondition Loops (while)

Below is the general form of a **while** loop.

```
while(<condition>)
   <statement>
   <statement>
      etc
end
```



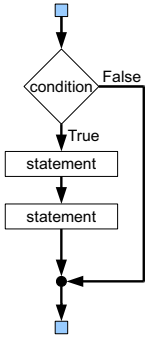
Example

```
A = 7;      %Find the square root of A.
x=1;        %First guess
err=1;      %Set error to get started

% Newton Raphson Iteration
while(err>0.0001)
    x = (x.^2+A)./(2*x);
    % calculate the error
    err = abs(A-x.^2);
end

disp(x);
```


Conditional Execution (if)

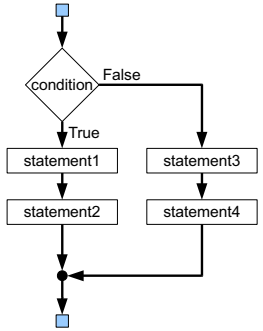
General form of a simple **if** statement.

```
If <condition>
  <statement>
  <statement>
     etc
end
```



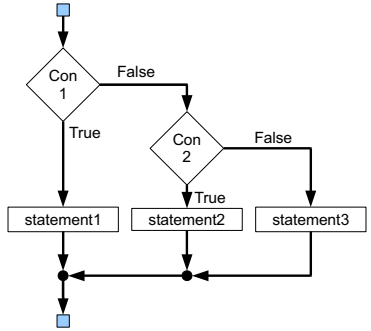
The form of **if** statement with **else**

```
if  <condition>
   <statement1>
   <statement2>
      etc
else
   <statement3>
   <statement4>
      etc
end
```



You can also have as many **elseif** parts as you like.

```
if <condition1>
    <statement1>
       etc
 elseif <condition2>
    <statement2>
       etc
 else
    <statement3>
       etc
 end
```



Switch

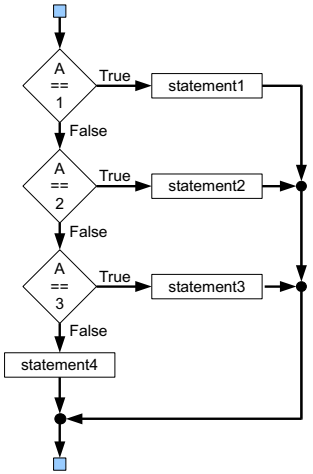
Execution depends on the value of a variable.

```
switch <variable>
 case 1
   <statement1>
      etc
 case 2
   <statement2>
      etc
 case 3
   <statement3>
      etc
 otherwise
   <statement4>
      etc
end
```

The case values can be any value that the switch variable can take. You can also put multiple values after the case.

```
case {1,2,3,4,5}
   <statement>
      etc
```

On the right A is the switch variable.



Functions

Function Definition	Using the Function

`function sum = myadd(a,b)`
 	
`% The first block of comments`
`% defines what is printed out when`
`% you type  help myadd`
 	
`% Comment not part of help`
`sum = a + b;`
	s = myadd(<expression>,<expression>)

`function [sum,diff] = add_sub(a,b)`
 	
`sum = a + b;`
`diff = a - b;`
	[s,d] = add_sub(<expression>,<expression>)

Introduction à Matlab

1 Introduction

Matlab *(MATrix LABoratory)* est un logiciel pour le calcul scientifique, orienté vers le vecteurs et les listes de données. Matlab est un langage interprété, chaque ligne d'un programme Matlab est lue, interprétée et exécutée.

Pour entrer dans Matlab cliquez sur Start/Programmes/Matlab sous l'environnement Windows, ou tapez matlab au prompt de la shell sous Unix. Matlab se présente avec un environnement interactif et un prompt (généralement >>) dans lequel on peut introduire des *commandes*. Par exemple, pour sortir de Matlab tapez la commande:

```
>> quit
```

2 Aide en ligne

Matlab offre une aide en ligne très complète. Tapez:

```
>> help commande
```

pour une explication de l'utilisation de la commande Matlab `commande`

```
>> lookfor sujet
```

pour une liste des commandes Matlab qui concernent le sujet `sujet`

```
>> helpwin
```

pour ouvrir un guide des commandes Matlab sur différents sujets.

3 Déclaration des variables

Matlab permet de créer et d'initialiser des variables. La déclaration des variables en Matlab suit les règles suivantes:

- toutes les variables sont des *matrices*
- pas de déclaration de type

`>> a=5`	*variable scalaire*	(1×1)
`>> b=[4  6]`	*vecteur ligne*	(1×2)
`>> c=[-5; 2]`	*vecteur colonne*	(2×1)
`>> d=[2 3; -1 7]`	*matrice carrée*	(2×2)

Dans ces exemples on a utilisé les opérateurs suivants:

- séparateur de ligne: point virgule ou return
- séparateur de colonne: virgule ou espace blanc

Il faut noter que Matlab montre les nombres avec quatre chiffres après la virgule, tandis que la représentation interne est faite avec 16 chiffres. Pour changer la façon de montrer les nombres en Matlab, on utilise la commande `format`. Par exemple, si avant de taper `>> pi` (en Matlab la variable `pi` est une approximation de π), nous écrivons

`>> format long`	nous obtiendrons	`3.1416`;
`>> format short`	nous obtiendrons	`3.14159265358979`;
`>> format long e`	nous obtiendrons	`3.141592653589793e+00`;
`>> format short e`	nous obtiendrons	`3.1416e+00`.

4 Workspace

Matlab permet de connaître plusieurs informations sur les variables déclarées. Tapez:

`>> who`	pour afficher toutes les variables.
`>> whos`	pour afficher toutes les variables avec indication sur leur taille.
`>> size(a)`	pour afficher les dimensions de la matrice `a`.
`>> clear var`	pour effacer la variable `var`.
`>> clear` (ou `>> clear all`)	pour effacer toutes les variables.

Pour ces opérations vous pouvez aussi utiliser l'interface graphique de Matlab.

5 Opérations fondamentales

Matlab peut effectuer plusieurs opérations entre matrices. Les opérations fondamentales peuvent être partagées en deux catégories.

Opérations matricielles

Les opérations matricielles usuelles sont définies par `+, -, *, /, ^`

`>> C = A + B`	est la somme matricielle, $C_{ij} = A_{ij} + B_{ij}$
`>> C = A * B`	est le produit matriciel, $C_{ij} = \sum_k A_{ik} B_{kj}$
`>> C = A / B`	est la division matricielle, $C = AB^{-1}$
`>> C = A^3`	est la troisième puissance matricielle (`C = A*A*A`)

Il faut remarquer que cettes opérations sont bien définies seulement si les matrices ont des *dimensions cohérentes*. Pour l'addition `A+B`, `A` et `B` doivent avoir la même taille; pour le produit `A*B`, le nombre des colonnes de `A` et le nombre des lignes de `B` doivent être égaux; les opérations `A/B` et `B^3` demandent une matrice `B` carrée. Par exemple:

```
>> A = [1 2 3; 4 5 6]; B = [7 8 9; 10 11 12]; C = [13 14; 15 16; 17 18];
>> A + B
ans =

     8    10    12
    14    16    18

>> A + C
```



```
??? Error using ==> +
Matrix dimensions must agree.

>> A * C
ans =

    94   100
   229   244

>> A * B
??? Error using ==> *
Inner matrix dimensions must agree.
```

Notez que Matlab renvoie un message d'erreur si le dimensions des matrices ne s'accordent pas avec l'opération commandée.

Opérations élément par élément

Pour exécuter des opérations entre matrices élément par élément il faut faire précéder l'opérateur d'un point. Les opérateurs élément par élément sont donc `.* ./ .^`

`>> C = A .* B`	est le produit élément par élément, $C_{ij} = A_{ij} B_{ij}$
`>> C = A ./ B`	est la division élément par élément, $C_{ij} = \frac{A_{ij}}{B_{ij}}$
`>> C = A.^3`	est la troisième puissance élément par élément, $C_{ij} = A_{ij}^3$

6 Manipulation des matrices

Après avoir défini la matrice `A` (par exemple la matrice carrée `A` de taille `n`), Matlab permet les opérations suivantes sur les *sous-matrices* de `A`.

Extraction de sous-matrices

`>> A(2,3)`	extraction de l' élément A_{23}
`>> A(:,5)`	extraction de la colonne $[A_{13}; \ldots; A_{n3}]$
`>> A(1:4,3)`	extraction de la sous-colonne $[A_{13}; \ldots; A_{43}]$
`>> A(1,:)`	extraction de la ligne $[A_{1j}, \ldots, A_{1n}]$
`>> diag(A)`	extraction de la diagonale $[A_{11}; \ldots; A_{nn}]$

Construction de matrices particulières

Matlab permet, en plus, de définir des matrices particulières (une fois définis les entiers `n`, `m` et le vecteur `v`).

`>> A = eye(n)`	matrice identité $n \times n$
`>> A = diag(v)`	matrice diagonale avec `v` comme diagonale
`>> A = zeros(n,m)`	matrice de zéros avec `n` lignes et `m` colonnes
`>> A = ones(n,m)`	matrice de un avec `n` lignes et `m` colonnes
`>> A = rand(n,m)`	matrice aléatoire avec `n` lignes et `m` colonnes
`>> x = [but:pas:fin]`	vecteur ligne de points équidistribués avec pas `pas` entre `but` et `fin`

Fonctions matricielles

Soit `A` une matrice, Matlab permet d'effectuer directement les opérations suivantes.

`>> C = A'`	transposée de `A`, $C_{ij} = A_{ji}$
`>> C = inv(A)`	inverse de `A` (matrice carrée), $C = A^{-1}$
`>> d = det(A)`	déterminant de `A` (matrice carrée)
`>> r = rank(A)`	rang de `A`
`>> nrm = norm(A)`	norme 2 de `A`
`>> v = eig(A)`	valeurs propres de `A` (matrice carrée)

Soit `A` une matrice carrée de taille `n` et soit `b` un vecteur de taille `n`, alors le vecteur `x`, solution du système linéaire `Ax = b` est défini par

```
>> x = A^-1 *  b
```

Cette commande est théoriquement correcte, cependant si on est intéressé seulement à la solution `x` et pas à l'inverse `A^-1`, il est préférable utiliser un *backslash*

```
>> x = A \ b
```

Cette commande résout le système linéaire avec des algorithmes fiables et optimales.

7 Graphisme 2D

Matlab offre plusieurs possibilités pour tracer un graphe en 2D. On va en présenter deux: la commande `plot` et la commande `fplot`.
Avant d'expliquer ces deux commandes en détail, on souligne qu'avec `plot` on doit toujours utiliser des vecteurs, alors qu'avec `fplot` non.
On considère maintenant la fonction $f(x) = x^3 - 2\sin x + 1$ et on voit comment on peut tracer son graphe dans l'intervalle $[-1, 1]$, en utilisant les commandes `plot` et `fplot`.

Commande `plot`

Pour tracer le graphe de $f(x)$ il faut passer par les étapes suivantes :

- Définir la fonction $f(x)$:

  ```
  >> f = 'x.^3 - 2*sin(x) + 1';
  ```

  le mot `f` contient la définition de la fonction choisie (les guillemets font partie de la définition dans la syntaxe Matlab).

- Définir un *vecteur de points* dans l'intervalle donné:

  ```
  >> x = [-1:0.1:+1];
  ```

  on a défini un vecteur de 21 points equidistribués dans l'intervalle donné, avec un pas de 0.1.

- Évaluer la fonction `f` dans l'intervalle $[-1, 1]$, en utilisant la commande `eval`:

  ```
  >> y = eval(f);
  ```

  on note que dans la définition de la fonction f on a dû écrire `.^` (puissance élément par élément) et pas `^` (puissance matricielle), parce qu'il faut évaluer la fonction en chaque élément du *vecteur* `x`. C'est-à-dire:

  $$y(i) = x(i)^3 - 2\sin(x(i)) + 1, \quad \text{pour } i = 1, \ldots, 21;$$

  La variable `y` est donc un *vecteur* qui contient les valeurs de `f` pour chaque entrée du vecteur `x`.

- Tracer le graphe :

  ```
  >> plot(x,y); grid;
  ```

  cette commande ouvre une fenêtre avec le graphe. La commande `grid` dessine une grille de repère.

Le résultat de ces commandes est représenté dans la Figure 1.

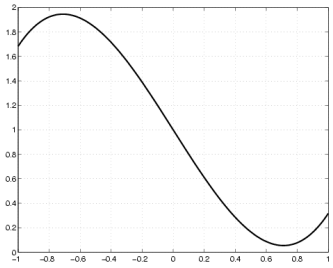


Figure 1: Graphe de la fonction $f(x) = x^3 - 2\sin(x) + 1$

Commande `fplot`

Pour tracer le graphe avec la commande `fplot` il faut passer par les étapes suivantes :

- Définir la fonction $f(x)$:

  ```
  >> f = 'x^3 - 2*sin(x) + 1';
  ```

- Tracer le graphe :

  ```
  >> fplot(f,[-1,1]); grid;
  ```

  La commande `fplot` demande la définition de la fonction `f` et de l'intervalle où il faut la tracer. La définition des vecteurs `x` et `y` *n'est plus nécessaire* et on note que dans la définition de `f` il suffit d'écrire `f='x^3 - ...'` au lieu de `f = x.^3 - ...'`, comme on avait fait pour utiliser `plot`. Le graphe obtenu est encore montré dans la Figure 1.

Exemple Tracer le graphe des fonctions $f_1(x) = \sin(2x) - 1 + x$ et $f_2(x) = x^3 - 2\sin(x)$ dans l'intervalle $[-1, 1]$ sur la même fenêtre.

En utilisant la commande `fplot`, il faut procéder de la façon suivante:

- Définir les fonctions:

  ```
  >> f1 = 'sin(2*x) - 1 + x';
  >> f2 = 'x^3 - 2*sin(x)';
  ```

- Tracer le graphe:

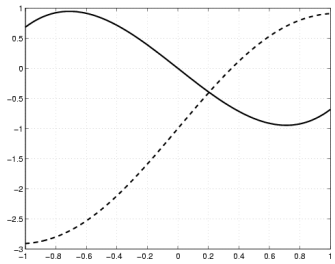


Figure 2: Résultat de l'exemple

  ```
  >> fplot(f1,[-1,1]);  grid;  hold on;
  ```

  La commande `hold on` affiche tous les prochains graphes sur la fenêtre qu'on vient d'ouvrir. Donc le graphe:

  ```
  >> fplot(f2,[-1,1]);
  ```

  est affiché sur la même fenêtre, comme on peut le voir dans la Figure 2.
  La commande `hold off` arrête cet effet.

Pour plus de détails sur les fonctions `plot` et `fplot` (par exemple comment changer la couleur du graphe) tapez `help plot` ou `help fplot`.

8 Boucles de contrôle

Matlab offre, comme plusieurs autres langages, quelques boucles de contrôle. Cettes commandes sont spécialement utiles pour écrire des programmes Matlab.

Boucle `for`

Si l'on veut exécuter des instructions pour chaque valeur

$$i = m,\ m+1,\ \ldots,\ n$$

d'une certaine variable i entre deux bornes assignés m et n, on utilise `for`. Par exemple pour calculer le produit scalaire `ps` entre deux vecteurs `x` et `y` de taille `n` on utilise:

```
>> ps = 0;
>> for i = 1:n;
>>    ps = ps + x(i)*y(i);
>> end;
```

Cette boucle est équivalente au le produit matriciel entre le vecteur transposé de `x` et le vecteur `y`, en supposant que les facteurs soient deux vecteurs colonne:

```
>> ps = x'*y;
```

Boucle `while`

Si l'on veut exécuter plusieurs fois des instructions tandis qu'une expression logique est vraie, on utilise `while`. Par exemple, le même calcul qu'on a considéré avec la boucle `for` peut être exécuté avec

```
>> ps = 0;
>> i = 0;
>> while (i < n);
>>    i = i + 1;
>>    ps = ps + x(i)*y(i);
>> end;
```

Instruction conditionelle `if`

Si l'on veut exécuter des instructions seulement si une expression logique est vraie, on utilise `if`. Par exemple, si l'on veut calculer la racine carrée d'une variable scalaire `r` seulement si `r` n'est pas négative:

```
>> if (r >= 0)
>>   racine = sqrt(r);
>> end;
```

On a plusieurs *opérateurs logiques* à disposition:

Opérateur	Action logique
`&`	*and*
`|`	*or*
`~`	*not*
`==`	*equal to*

Taper par exemple `help &` pour une explication de la liste des opérateurs.

9 Scripts et fonctions

Matlab permet d'écrire des programmes. Les outils principaux sont les script files et les fonctions.

Script files

Un script file est une suite de commandes Matlab. Les noms des script files doivent avoir l'extension "`.m`". Pour exécuter un script file tapez son nom, sans extension, dans le prompt Matlab.

Fonctions

Une fonction Matlab est une suite de commandes qui nécessite un input pour être exécutée et qui renvoie un output. La déclaration des fonctions en Matlab suit les règles suivantes.

- Une fonction est contenue dans un fichier "`.m`" avec le même nom que la fonction.
- Le fichier qui définit une fonction doit commencer par:
  `function` [*output arguments*] = `nom_fonction(`*input arguments*`)`
  Par exemple, l'en-tête d'une fonction pour la méthode de dichotomie pourrait être:

```
function [zero,res,niter]=bisection(fun,a,b,tol,nmax,varargin)
%BISECTION Find function zeros.
%   ZERO=BISECTION(FUN,A,B,TOL,NMAX) tries to find a zero ZERO
%   of the continuous function FUN in the interval [A,B]
%   using the bisection method.
            .
            .
            .  % istructions
            .
 zero = ...; res = ...; niter = ...;
            .
            .  % istructions
            .
            .

return        % fin du corps de la fonction
```

- Les lignes de commentaires qui éventuellement suivent cet en-tête constituent le help de la fonction. En Matlab les lignes de commentaire sont introduites par `%`.
- Toutes les variables internes à la fonction sont locales.

Par exemple, supposons qu'on aie écrit sur le fichier `my_function.m` le lignes suivantes:

```
function f = my_function(x);
f = x.^3 - 2*sin(x) + 1;
return;
```

Il est possible d'utiliser la fonction `my_function.m` de la même façon que les autres commandes Matlab. Donc les commandes

```
>> x = 0;
>> y = x.^3 - 2*sin(x) + 1
```

et

```
>> x = 0;
>> y = my_function(x)
```

sont équivalentes et renvoient le même résultat:

```
y =
     1
```

Fonctions de plusieurs variables

Pour définir des fonctions dépendantes d'un ou plusieurs paramètres nous pouvons utiliser les fonctions *inline*. Pour définir une fonction *inline* `f` qui dépend de la variable `x` on écrit:

```
>> f = inline('log(x) - 1','x');
```

Si `f` dépend de de plusieurs variables, nous écrirons par example

```
>> f = inline('log(x) - p','x','p');
```

Pour évaluer une fonction `f` définie grâce à *inline*, il faut utiliser la commande `feval` en donnant les valeurs des paramètres comme montré dans l'exemple suivant:

```
>> f = inline('x.^3 + a.*x','x','a');
>> xval = 1; aval = 2;
>> fval = feval(f, xval, yval)
val =
     3
```

10 Polynômes en Matlab

Considérons un polynôme de degré n:

$$f(x) = a_n x^n \ + \ + a_{n-1} x^{n-1} \ + \ \ldots \ + a_1 x \ + \ a_0.$$

Matlab répresente les polynômes de degré n sous la forme d'un vecteur $p = [a_n, a_{n-1}, \ldots, a_0]$ de taille $n+1$ qui contient les coefficients en ordre décroissante par rapport au degré associé. Par exemple, le vecteur associé au polynôme $f(x) = 3x^3 - 4x^2 + x$ est

```
>> p = [3, -4, 1, 0];
```


Mois	Heure de son réveil (moyenne)	Mois	Heure de son réveil (moyenne)
1	8.0	7	8.0
2	7.5	8	10.0
3	7.5	9	8.0
4	7.5	10	7.5
5	7.6	11	7.5
6	7.8	12	7.7

Table 1: Heure de son réveil (de 0 à 24 avec décimes d'heure)

Pour évaluer ce polynôme on utilise la commande `polyval`:

```
>> x = [0:.1:1];
>> y = polyval(p, x);
```

Dans ce cas, les éléments du vecteur `y` sont les valeurs du polynôme calculées pour chaque élément du vecteur `x`.
Pour obtenir le vecteur associé au polynôme de degré `n` approximant un jeu de données on utilise `polyfit`. Si la taille des données est plus grande de `n+1` on a le polynôme approximant au sens des moindres carrés; si elle est égal à `n+1` on a le polynôme interpolant. Par exemple, si on veut calculer le polynôme de degré 8 qui approxime le données de la Table 10 (au sens des moindre carrées) et en tracer le graphe, il faut utiliser les commandes qui suivent;

```
x = [1 2 3 4 5 6 7 8 9 10 11 12];
y = [8 7.5 7.5 7.5 7.6 7.8 8 10 8 7.5 7.5 7.7];
p = polyfit(x,y,8);   % Calculons le polynome interpolant (degre=8)
plot(x, y, 'or');
axis([1 12 6 12])
hold on;
f = inline('polyval(p,x)','x','p');
fplot(f, [1 12], [], [], [], p);  % Trace le graphe de f(x,p)
                                  % pour x entre 0 et 12, en donnant
                                  % p comme parametre de f.
```

Il faut noter que `f` est déclarée comme fonction *inline* avec les arguments `x` et `p` (le vecteur associé au polynôme interpolant); la commande `fplot` (voir `help fplot`) accepte le paramètre `p` à donner à `f`.

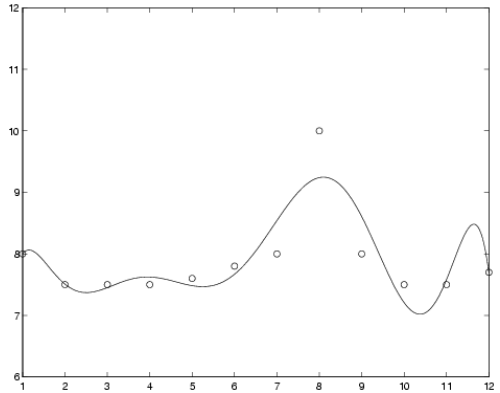


Figure 3: Données et polynôme d'interpolation

MATLAB Primer

Third Edition

Kermit Sigmon
Department of Mathematics
University of Florida

Department of Mathematics • University of Florida • Gainesville, FL 32611
sigmon@math.ufl.edu

On the Third Edition

The Third Edition of the MATLAB Primer is based on version 4.0/4.1 of MATLAB. While this edition reflects an extensive general revision of the Second Edition, most significant is the new information to help one begin to use the major new features of version 4.0/4.1, the sparse matrix and enhanced graphics capabilities.

The plain TEX source and corresponding PostScript file of the latest printing of the MATLAB Primer are always available via anonymous ftp from:

Address: `math.ufl.edu` Directory: `pub/matlab` Files: `primer.tex, primer.ps`

You are advised to download anew each term the latest printing of the Primer since minor improvements and corrections may have been made in the interim. If ftp is unavailable to you, the Primer can be obtained via `listserv` by sending an email message to `listserv@math.ufl.edu` containing the single line `send matlab/primer.tex`.

Also available at this ftp site are both English (`primer35.tex, primer35.ps`) and Spanish (`primer35sp.tex, primer35sp.ps`) versions of the Second Edition of the Primer, which was based on version 3.5 of MATLAB. The Spanish translation is by Celestino Montes, University of Seville, Spain. A Spanish translation of the Third Edition is under development.

Users of the Primer usually appreciate the convenience and durability of a bound copy with a cover, copy center style.

(12-93)

Department of Mathematics • University of Florida • Gainesville, FL 32611
sigmon@math.ufl.edu

Introduction

MATLAB is an interactive, matrix-based system for scientific and engineering numeric computation and visualization. You can solve complex numerical problems in a fraction of the time required with a programming language such as Fortran or C. The name MATLAB is derived from MATrix LABoratory.

The purpose of this Primer is to help you begin to use MATLAB. It is not intended to be a substitute for the User's Guide and Reference Guide for MATLAB. The Primer can best be used hands-on. You are encouraged to work at the computer as you read the Primer and freely experiment with examples. This Primer, along with the on-line help facility, usually suffice for students in a class requiring use of MATLAB.

You should liberally use the on-line help facility for more detailed information. When using MATLAB, the command `help` *functionname* will give information about a specific function. For example, the command `help eig` will give information about the eigenvalue function `eig`. By itself, the command `help` will display a list of topics for which on-line help is available; then `help` *topic* will list those specific functions under this topic for which help is available. The list of functions in the last section of this Primer also gives most of this information. You can preview some of the features of MATLAB by first entering the command `demo` and then selecting from the options offered.

The scope and power of MATLAB go far beyond these notes. Eventually you will want to consult the MATLAB User's Guide and Reference Guide. Copies of the complete documentation are often available for review at locations such as consulting desks, terminal rooms, computing labs, and the reserve desk of the library. Consult your instructor or your local computing center to learn where this documentation is located at your institution.

MATLAB is available for a number of environments: Sun/Apollo/VAXstation/HP workstations, VAX, MicroVAX, Gould, PC and AT compatibles, 80386 and 80486 computers, Apple Macintosh, and several parallel machines. There is a relatively inexpensive Student Edition available from Prentice Hall publishers. The information in these notes applies generally to all of these environments.

MATLAB is licensed by The MathWorks, Inc., 24 Prime Park Way, Natick, MA 01760, (508)653-1415, Fax: (508)653-2997, Email: info@mathworks.com.

Contents

Page

1. Accessing MATLAB 1
2. Entering matrices 1
3. Matrix operations, array operations 2
4. Statements, expressions, variables; saving a session 3
5. Matrix building functions 4
6. For, while, if — and relations 4
7. Scalar functions 7
8. Vector functions 7
9. Matrix functions 7
10. Command line editing and recall 8
11. Submatrices and colon notation 8
12. M-files: script files, function files 9
13. Text strings, error messages, input 12
14. Managing M-files 13
15. Comparing efficiency of algorithms: flops, tic, toc 14
16. Output format 14
17. Hard copy 15
18. Graphics 15
    planar plots (15), hardcopy (17), 3-D line plots (18)
    mesh and surface plots (18), Handle Graphics (20)
19. Sparse matrix computations 20
20. Reference 22

1. Accessing MATLAB.

On most systems, after logging in one can enter MATLAB with the system command `matlab` and exit MATLAB with the MATLAB command `quit` or `exit`. However, your local installation may permit MATLAB to be accessed from a menu or by clicking an icon.

On systems permitting multiple processes, such as a Unix system or MS Windows, you will find it convenient, for reasons discussed in section 14, to keep both MATLAB and your local editor active. If you are working on a platform which runs processes in multiple windows, you will want to keep MATLAB active in one window and your local editor active in another.

You should consult your instructor or your local computer center for details of the local installation.

2. Entering matrices.

MATLAB works with essentially only one kind of object—a rectangular numerical matrix with possibly complex entries; all variables represent matrices. In some situations, 1-by-1 matrices are interpreted as scalars and matrices with only one row or one column are interpreted as vectors.

Matrices can be introduced into MATLAB in several different ways:

- Entered by an explicit list of elements,
- Generated by built-in statements and functions,
- Created in a diskfile with your local editor,
- Loaded from external data files or applications (see the User's Guide).

For example, either of the statements

```
A = [1 2 3; 4 5 6; 7 8 9]
```

and

```
A = [
1 2 3
4 5 6
7 8 9 ]
```

creates the obvious 3-by-3 matrix and assigns it to a variable A. Try it. The elements within a row of a matrix may be separated by commas as well as a blank. When listing a number in exponential form (e.g. 2.34e-9), blank spaces must be avoided.

MATLAB allows complex numbers in all its operations and functions. Two convenient ways to enter complex matrices are:

```
A = [1 2;3 4] + i*[5 6;7 8]
A = [1+5i 2+6i;3+7i 4+8i]
```

When listing complex numbers (e.g. 2+6i) in a matrix, blank spaces must be avoided. Either i or j may be used as the imaginary unit. If, however, you use i and j as variables and overwrite their values, you may generate a new imaginary unit with, say, `ii = sqrt(-1)`.

Listing entries of a large matrix is best done in an ASCII file with your local editor, where errors can be easily corrected (see sections 12 and 14). The file should consist of a rectangular array of just the numeric matrix entries. If this file is named, say, `data.ext` (where `.ext` is any extension), the MATLAB command `load data.ext` will read this file to the variable *data* in your MATLAB workspace. This may also be done with a script file (see section 12).

The built-in functions `rand`, `magic`, and `hilb`, for example, provide an easy way to create matrices with which to experiment. The command `rand(n)` will create an $n \times n$ matrix with randomly generated entries distributed uniformly between 0 and 1, while `rand(m,n)` will create an $m \times n$ one. `magic(n)` will create an integral $n \times n$ matrix which is a magic square (rows, columns, and diagonals have common sum); `hilb(n)` will create the $n \times n$ Hilbert matrix, the king of ill-conditioned matrices (m and n denote, of course, positive integers). Matrices can also be generated with a for-loop (see section 6 below).

Individual matrix and vector entries can be referenced with indices inside parentheses in the usual manner. For example, $A(2,3)$ denotes the entry in the second row, third column of matrix A and $x(3)$ denotes the third coordinate of vector x. Try it. A matrix or a vector will only accept *positive* integers as indices.

3. Matrix operations, array operations.

The following matrix operations are available in MATLAB:

$+$	addition
$-$	subtraction
$*$	multiplication
$\hat{}$	power
$'$	conjugate transpose
$\backslash$	left division
$/$	right division

These matrix operations apply, of course, to scalars (1-by-1 matrices) as well. If the sizes of the matrices are incompatible for the matrix operation, an error message will result, except in the case of scalar-matrix operations (for addition, subtraction, and division as well as for multiplication) in which case each entry of the matrix is operated on by the scalar.

The "matrix division" operations deserve special comment. If A is an invertible square matrix and b is a compatible column, resp. row, vector, then

$x = A\backslash b$ is the solution of $A * x = b$ and, resp.,
$x = b/A$ is the solution of $x * A = b$.

In left division, if A is square, then it is factored using Gaussian elimination and these factors are used to solve $A * x = b$. If A is not square, it is factored using Householder orthogonalization with column pivoting and the factors are used to solve the under- or over- determined system in the least squares sense. Right division is defined in terms of left division by $b/A = (A'\backslash b')'$.

Array operations.

The matrix operations of addition and subtraction already operate entry-wise but the other matrix operations given above do not—they are *matrix* operations. It is important to observe that these other operations, $*$, `^`, \, and /, can be made to operate entry-wise by preceding them by a period. For example, either `[1,2,3,4].*[1,2,3,4]` or `[1,2,3,4].^2` will yield `[1,4,9,16]`. Try it. This is particularly useful when using Matlab graphics.

4. Statements, expressions, and variables; saving a session.

MATLAB is an *expression* language; the expressions you type are interpreted and evaluated. MATLAB statements are usually of the form

variable = *expression*, or simply
expression

Expressions are usually composed from operators, functions, and variable names. Evaluation of the expression produces a matrix, which is then displayed on the screen and assigned to the variable for future use. If the variable name and = sign are omitted, a variable `ans` (for answer) is automatically created to which the result is assigned.

A statement is normally terminated with the carriage return. However, a statement can be continued to the next line with three or more periods followed by a carriage return. On the other hand, several statements can be placed on a single line if separated by commas or semicolons.

If the last character of a statement is a semicolon, the printing is suppressed, but the assignment is carried out. This is essential in suppressing unwanted printing of intermediate results.

MATLAB is case-sensitive in the names of commands, functions, and variables. For example, `solveUT` is not the same as `solveut`.

The command `who` (or `whos`) will list the variables currently in the workspace. A variable can be cleared from the workspace with the command `clear` *variablename*. The command `clear` alone will clear all nonpermanent variables.

The permanent variable `eps` (epsilon) gives the machine unit roundoff—about 10^{-16} on most machines. It is useful in specifying tolerences for convergence of iterative processes.

A runaway display or computation can be stopped on most machines without leaving MATLAB with CTRL-C (CTRL-BREAK on a PC).

Saving a session.

When one logs out or exits MATLAB all variables are lost. However, invoking the command `save` before exiting causes all variables to be written to a non-human-readable diskfile named `matlab.mat`. When one later reenters MATLAB, the command `load` will restore the workspace to its former state.

5. Matrix building functions.

Convenient matrix building functions are

eye	identity matrix
zeros	matrix of zeros
ones	matrix of ones
diag	create or extract diagonals
triu	upper triangular part of a matrix
tril	lower triangular part of a matrix
rand	randomly generated matrix
hilb	Hilbert matrix
magic	magic square
toeplitz	see `help toeplitz`

For example, `zeros(m,n)` produces an m-by-n matrix of zeros and `zeros(n)` produces an n-by-n one. If A is a matrix, then `zeros(size(A))` produces a matrix of zeros having the same size as A.

If x is a vector, `diag(x)` is the diagonal matrix with x down the diagonal; if A is a square matrix, then `diag(A)` is a vector consisting of the diagonal of A. What is `diag(diag(A))`? Try it.

Matrices can be built from blocks. For example, if A is a 3-by-3 matrix, then

```
B = [A, zeros(3,2); zeros(2,3), eye(2)]
```

will build a certain 5-by-5 matrix. Try it.

6. For, while, if — and relations.

In their basic forms, these MATLAB flow control statements operate like those in most computer languages.

For.

For example, for a given n, the statement

```
x = []; for i = 1:n, x=[x,i^2], end
```

or

```
x = [];
for i = 1:n
   x = [x,i^2]
end
```

will produce a certain n-vector and the statement

```
x = []; for i = n:-1:1, x=[x,i^2], end
```

will produce the same vector in reverse order. Try them. Note that a matrix may be empty (such as `x = []`).

The statements

```
for i = 1:m
   for j = 1:n
      H(i, j) = 1/(i+j-1);
   end
end
H
```

will produce and print to the screen the m-by-n hilbert matrix. The semicolon on the inner statement is essential to suppress printing of unwanted intermediate results while the last `H` displays the final result.

The `for` statement permits *any* matrix to be used instead of `1:n`. The variable just consecutively assumes the value of each column of the matrix. For example,

```
s = 0;
for c = A
   s = s + sum(c);
end
```

computes the sum of all entries of the matrix A by adding its column sums (Of course, `sum(sum(A))` does it more efficiently; see section 8). In fact, since `1:n = [1,2,3,...,n]`, this column-by-column assigment is what occurs with "`if i = 1:n,...`" (see section 11).

While.

The general form of a `while` loop is

```
while relation
   statements
end
```

The statements will be repeatedly executed as long as the relation remains true. For example, for a given number a, the following will compute and display the smallest nonnegative integer n such that $2^n \geq a$:

```
n = 0;
while 2^n < a
   n = n + 1;
end
n
```

If.

The general form of a simple `if` statement is

```
if relation
   statements
end
```

The statements will be executed only if the relation is true. Multiple branching is also possible, as is illustrated by

```
if n < 0
   parity = 0;
elseif rem(n,2) == 0
   parity = 2;
else
   parity = 1;
end
```

In two-way branching the elseif portion would, of course, be omitted.

Relations.

The relational operators in MATLAB are

<	less than
>	greater than
<=	less than or equal
>=	greater than or equal
==	equal
~=	not equal.

Note that "=" is used in an assignment statement while "==" is used in a relation. Relations may be connected or quantified by the logical operators

&	and
|	or
~	not.

When applied to scalars, a relation is actually the scalar 1 or 0 depending on whether the relation is true or false. Try entering `3 < 5, 3 > 5, 3 == 5,` and `3 == 3`. When applied to matrices of the same size, a relation is a matrix of 0's and 1's giving the value of the relation between corresponding entries. Try `a = rand(5), b = triu(a), a == b`.

A relation between matrices is interpreted by `while` and `if` to be true if each entry of the relation matrix is nonzero. Hence, if you wish to execute *statement* when matrices A and B are equal you could type

```
if A == B
   statement
end
```

but if you wish to execute *statement* when A and B are not equal, you would type

```
if any(any(A ~= B))
   statement
end
```

or, more simply,

```
if A == B else
   statement
end
```

Note that the seemingly obvious

```
if A ~= B, statement, end
```

will not give what is intended since *statement* would execute only if *each* of the corresponding entries of A and B differ. The functions `any` and `all` can be creatively used to reduce matrix relations to vectors or scalars. Two `any`'s are required above since `any` is a vector operator (see section 8).

7. Scalar functions.

Certain MATLAB functions operate essentially on scalars, but operate element-wise when applied to a matrix. The most common such functions are

sin	asin	exp	abs	round
cos	acos	log (natural log)	sqrt	floor
tan	atan	rem (remainder)	sign	ceil

8. Vector functions.

Other MATLAB functions operate essentially on a vector (row or column), but act on an m-by-n matrix ($m \geq 2$) in a column-by-column fashion to produce a row vector containing the results of their application to each column. Row-by-row action can be obtained by using the transpose; for example, `mean(A')'`. A few of these functions are

max	sum	median	any
min	prod	mean	all
sort		std	

For example, the maximum entry in a matrix A is given by `max(max(A))` rather than `max(A)`. Try it.

9. Matrix functions.

Much of MATLAB's power comes from its matrix functions. The most useful ones are

eig	eigenvalues and eigenvectors
chol	cholesky factorization
svd	singular value decomposition
inv	inverse
lu	LU factorization
qr	QR factorization
hess	hessenberg form
schur	schur decomposition
rref	reduced row echelon form
expm	matrix exponential
sqrtm	matrix square root
poly	characteristic polynomial
det	determinant
size	size
norm	1-norm, 2-norm, F-norm, ∞-norm
cond	condition number in the 2-norm
rank	rank

MATLAB functions may have single or multiple output arguments. For example,

`y = eig(A)`, or simply `eig(A)`

produces a column vector containing the eigenvalues of A while

`[U,D] = eig(A)`

produces a matrix U whose columns are the eigenvectors of A and a diagonal matrix D with the eigenvalues of A on its diagonal. Try it.

10. Command line editing and recall.

The command line in MATLAB can be easily edited. The cursor can be positioned with the left/right arrows and the Backspace (or Delete) key used to delete the character to the left of the cursor. Other editing features are also available. On a PC try the Home, End, and Delete keys; on a Unix system or a PC the Emacs commands Ctl-a, Ctl-e, Ctl-d, and Ctl-k work; on other systems see `help cedit` or `type cedit`.

A convenient feature is use of the up/down arrows to scroll through the stack of previous commands. One can, therefore, recall a previous command line, edit it, and execute the revised command line. For small routines, this is much more convenient that using an M-file which requires moving between MATLAB and the editor (see sections 12 and 14). For example, flopcounts (see section 15) for computing the inverse of matrices of various sizes could be compared by repeatedly recalling, editing, and executing

```
a = rand(8); flops(0), inv(a); flops
```

If one wanted to compare plots of the functions $y = \sin mx$ and $y = \sin nx$ on the interval $[0, 2\pi]$ for various m and n, one might do the same for the command line:

```
m=2; n=3; x=0:.01:2*pi; y=sin(m*x); z=cos(n*x); plot(x,y,x,z)
```

11. Submatrices and colon notation.

Vectors and submatrices are often used in MATLAB to achieve fairly complex data manipulation effects. "Colon notation" (which is used both to generate vectors and reference submatrices) and subscripting by integral vectors are keys to efficient manipulation of these objects. Creative use of these features to vectorize operations permits one to minimize the use of loops (which slows MATLAB) and to make code simple and readable. *Special effort should be made to become familiar with them.*

The expression `1:5` (met earlier in `for` statements) is actually the row vector `[1 2 3 4 5]`. The numbers need not be integers nor the increment one. For example,

```
0.2:0.2:1.2
```

gives `[0.2, 0.4, 0.6, 0.8, 1.0, 1.2]`, and

`5:-1:1` gives `[5 4 3 2 1]`.

The following statements will, for example, generate a table of sines. Try it.

```
x = [0.0:0.1:2.0]';
y = sin(x);
[x y]
```

Note that since `sin` operates entry-wise, it produces a vector y from the vector x.

The colon notation can be used to access submatrices of a matrix. For example,

`A(1:4,3)` is the column vector consisting of the first four entries of the third column of A.

A colon by itself denotes an entire row or column:

`A(:,3)` is the third column of A, and `A(1:4,:)` is the first four rows.

Arbitrary integral vectors can be used as subscripts:

`A(:,[2 4])` contains as columns, columns 2 and 4 of A.

Such subscripting can be used on both sides of an assignment statement:

`A(:,[2 4 5]) = B(:,1:3)` replaces columns 2,4,5 of A with the first three columns of B. Note that the *entire* altered matrix A is printed and assigned. Try it.

Columns 2 and 4 of A can be multiplied on the right by the 2-by-2 matrix [1 2;3 4]:

```
A(:,[2,4]) = A(:,[2,4])*[1 2;3 4]
```

Once again, the entire altered matrix is printed and assigned.

If x is an n-vector, what is the effect of the statement `x = x(n:-1:1)`? Try it. Also try `y = fliplr(x)` and `y = flipud(x')`.

To appreciate the usefulness of these features, compare these MATLAB statements with a Pascal, FORTRAN, or C routine to effect the same.

12. M-files.

MATLAB can execute a sequence of statements stored in diskfiles. Such files are called "M-files" because they must have the file type of ".m" as the last part of their filename. Much of your work with MATLAB will be in creating and refining M-files. M-files are usually created using your local editor.

There are two types of M-files: *script files* and *function files*.

Script files.

A script file consists of a sequence of normal MATLAB statements. If the file has the filename, say, `rotate.m`, then the MATLAB command `rotate` will cause the statements in the file to be executed. Variables in a script file are global and will change the value of variables of the same name in the environment of the current MATLAB session.

Script files may be used to enter data into a large matrix; in such a file, entry errors can be easily corrected. If, for example, one enters in a diskfile `data.m`

```
A = [
1 2 3 4
5 6 7 8
];
```

then the MATLAB statement `data` will cause the assignment given in `data.m` to be carried out. However, it is usually easier to use the MATLAB function `load` (see section 2).

An M-file can reference other M-files, including referencing itself recursively.

Function files.

Function files provide extensibility to MATLAB. You can create new functions specific to your problem which will then have the same status as other MATLAB functions. Variables in a function file are by default local. A variable can, however, be declared global (see `help global`).

We first illustrate with a simple example of a function file.

```
function a = randint(m,n)
%RANDINT Randomly generated integral matrix.
%    randint(m,n) returns an m-by-n such matrix with entries
%    between 0 and 9.
a = floor(10*rand(m,n));
```

A more general version of this function is the following:

```
function a = randint(m,n,a,b)
%RANDINT Randomly generated integral matrix.
%    randint(m,n) returns an m-by-n such matrix with entries
%    between 0 and 9.
%    rand(m,n,a,b) return entries between integers a and b.
if nargin < 3, a = 0; b = 9; end
a = floor((b-a+1)*rand(m,n)) + a;
```

This should be placed in a diskfile with filename `randint.m` (corresponding to the function name). The first line declares the function name, input arguments, and output arguments; without this line the file would be a script file. Then a MATLAB statement `z = randint(4,5)`, for example, will cause the numbers 4 and 5 to be passed to the variables m and n in the function file with the output result being passed out to the variable z. Since variables in a function file are local, their names are independent of those in the current MATLAB environment.

Note that use of `nargin` ("number of input arguments") permits one to set a default value of an omitted input variable—such as a and b in the example.

A function may also have multiple output arguments. For example:

```
function [mean, stdev] = stat(x)
% STAT Mean and standard deviation
%    For a vector x, stat(x) returns the mean of x;
%    [mean, stdev] = stat(x) both the mean and standard deviation.
%    For a matrix x, stat(x) acts columnwise.
[m n] = size(x);
if m == 1
   m = n; % handle case of a row vector
end
mean = sum(x)/m;
stdev = sqrt(sum(x.^2)/m - mean.^2);
```

Once this is placed in a diskfile `stat.m`, a MATLAB command `[xm, xd] = stat(x)`, for example, will assign the mean and standard deviation of the entries in the vector x to

xm and xd, respectively. Single assignments can also be made with a function having multiple output arguments. For example, `xm = stat(x)` (no brackets needed around xm) will assign the mean of x to xm.

The `%` symbol indicates that the rest of the line is a comment; MATLAB will ignore the rest of the line. Moreover, the first few contiguous comment lines, which document the M-file, are available to the on-line help facility and will be displayed if, for example, `help stat` is entered. Such documentation should *always* be included in a function file.

This function illustrates some of the MATLAB features that can be used to produce efficient code. Note, for example, that `x.^2` is the matrix of squares of the entries of x, that `sum` is a vector function (section 8), that `sqrt` is a scalar function (section 7), and that the division in `sum(x)/m` is a matrix-scalar operation. Thus all operations are vectorized and loops avoided.

If you can't vectorize some computations, you can make your `for` loops go faster by preallocating any vectors or matrices in which output is stored. For example, by including the second statement below, which uses the function `zeros`, space for storing E in memory is preallocated. Without this MATLAB must resize E one column larger in each iteration, slowing execution.

```
M = magic(6);
E = zeros(6,50);
for j = 1:50
   E(:,j) = eig(M^i);
end
```

Some more advanced features are illustrated by the following function. As noted earlier, some of the input arguments of a function—such as *tol* in this example, may be made optional through use of `nargin` ("number of input arguments"). The variable `nargout` can be similarly used. Note that the fact that a relation is a number (1 when true; 0 when false) is used and that, when `while` or `if` evaluates a relation, "nonzero" means "true" and 0 means "false". Finally, the MATLAB function `feval` permits one to have as an input variable a string naming another function. (Also see `eval`.)

```
function [b, steps] = bisect(fun, x, tol)
%BISECT Zero of a function of one variable via the bisection method.
%    bisect(fun,x) returns a zero of the function.  fun is a string
%    containing the name of a real-valued MATLAB function of a
%    single real variable; ordinarily functions are defined in
%    M-files.  x is a starting guess.  The value returned is near
%    a point where fun changes sign.  For example,
%    bisect('sin',3) is pi.  Note the quotes around sin.
%
%    An optional third input argument sets a tolerence for the
%    relative accuracy of the result.  The default is eps.
%    An optional second output argument gives a matrix containing a
%    trace of the steps; the rows are of form [c f(c)].
```



```
% Initialization
if nargin < 3, tol = eps; end
trace = (nargout == 2);
if x ~= 0, dx = x/20; else, dx = 1/20; end
a = x - dx; fa = feval(fun,a);
b = x + dx; fb = feval(fun,b);

% Find change of sign.
while (fa > 0) == (fb > 0)
   dx = 2.0*dx;
   a = x - dx; fa = feval(fun,a);
   if (fa > 0) ~= (fb > 0), break, end
   b = x + dx; fb = feval(fun,b);
end
if trace, steps = [a fa; b fb]; end

% Main loop
while abs(b - a) > 2.0*tol*max(abs(b),1.0)
   c = a + 0.5*(b - a); fc = feval(fun,c);
   if trace, steps = [steps; [c fc]]; end
   if (fb > 0) == (fc > 0)
      b = c; fb = fc;
   else
      a = c; fa = fc;
   end
end
```

Some of MATLAB's functions are built-in while others are distributed as M-files. The actual listing of any non-built-in M-file—MATLAB's or your own—can be viewed with the MATLAB command `type` *functionname*. Try entering `type eig`, `type vander`, and `type rank`.

13. Text strings, error messages, input.

Text strings are entered into MATLAB surrounded by single quotes. For example,

```
s = 'This is a test'
```

assigns the given text string to the variable `s`.

Text strings can be displayed with the function `disp`. For example:

```
disp('this message is hereby displayed')
```

Error messages are best displayed with the function `error`

```
error('Sorry, the matrix must be symmetric')
```

since when placed in an M-File, it aborts execution of the M-file.

In an M-file the user can be prompted to interactively enter input data with the function `input`. When, for example, the statement

```
iter = input('Enter the number of iterations:  ')
```

is encountered, the prompt message is displayed and execution pauses while the user keys in the input data. Upon pressing the return key, the data is assigned to the variable `iter` and execution resumes.

14. Managing M-files.

While using MATLAB one frequently wishes to create or edit an M-file with the local editor and then return to MATLAB. One wishes to keep MATLAB active while editing a file since otherwise all variables would be lost upon exiting.

This can be easily done using the !-feature. If, while in MATLAB, you precede it with an !, any system command—such as those for editing, printing, or copying a file—can be executed without exiting MATLAB. If, for example, the system command `ed` accesses your editor, the MATLAB command

```
>> !ed rotate.m
```

will let you edit the file named `rotate.m` using your local editor. Upon leaving the editor, you will be returned to MATLAB just where you left it.

However, as noted in section 1, on systems permitting multiple processes, such as one running Unix or MS Windows, it may be preferable to keep both MATLAB and your local editor active, keeping one process suspended while working in the other. If these processes can be run in multiple windows, you will want to keep MATLAB active in one window and your editor active in another.

You should consult your instructor or your local computing center for details of the local installation.

Many debugging tools are available. See `help dbtype` or the list of functions in the last section.

When in MATLAB, the command `pwd` will return the name of the present working directory and `cd` can be used to change the working directory. Either `dir` or `ls` will list the contents of the working directory while the command `what` lists only the M-files in the directory. The MATLAB commands `delete` and `type` can be used to delete a diskfile and print an M-file to the screen, respectively. While these commands may duplicate system commands, they avoid the use of an !. You may enjoy entering the command `why` a few times.

M-files must be in a directory accessible to MATLAB. M-files in the present working directory are always accessible. On most mainframe or workstation network installations, personal M-files which are stored in a subdirectory of one's home directory named `matlab` will be accessible to MATLAB from any directory in which one is working. The current list of directories in MATLAB's search path is obtained by the command `path`. This command can also be used to add or delete directories from the search path. See `help path`.

15. Comparing efficiency of algorithms: flops, tic and toc.

Two measures of the efficiency of an algorithm are the number of floating point operations (flops) performed and the elapsed time.

The MATLAB function `flops` keeps a running total of the flops performed. The command `flops(0)` (not `flops = 0`!) will reset flops to 0. Hence, entering `flops(0)` immediately before executing an algorithm and `flops` immediately after gives the flop count for the algorithm. For example, the number of flops required to solve a given linear system via Gaussian elimination can be obtained with:

```
flops(0), x = A\b; flops
```

The elapsed time (in seconds) can be obtained with the stopwatch timers `tic` and `toc`; `tic` starts the timer and `toc` returns the elapsed time. Hence, the commands

`tic,` *any statement,* `toc`

will return the elapsed time for execution of the statement. The elapsed time for solving the linear system above can be obtained, for example, with:

```
tic, x = A\b; toc
```

You may wish to compare this time—and flop count—with that for solving the system using `x = inv(A)*b;`. Try it.

It should be noted that, on timesharing machines, elapsed time may not be a reliable measure of the efficiency of an algorithm since the rate of execution depends on how busy the computer is at the time.

16. Output format.

While all computations in MATLAB are performed in double precision, the format of the displayed output can be controlled by the following commands.

`format short`	fixed point with 4 decimal places (the default)
`format long`	fixed point with 14 decimal places
`format short e`	scientific notation with 4 decimal places
`format long e`	scientific notation with 15 decimal places
`format rat`	approximation by ratio of small integers
`format hex`	hexadecimal format
`format bank`	fixed dollars and cents
`format +`	+, -, blank

Once invoked, the chosen format remains in effect until changed.

The command `format compact` will suppress most blank lines allowing more information to be placed on the screen or page. The command `format loose` returns to the non-compact format. These commands are independent of the other format commands.

17. Hardcopy.

Hardcopy is most easily obtained with the **diary** command. The command

`diary` *filename*

causes what appears subsequently on the screen (except graphics) to be written to the named diskfile (if the filename is omitted it will be written to a default file named `diary`) until one gives the command `diary off`; the command `diary on` will cause writing to the file to resume, etc. When finished, you can edit the file as desired and print it out on the local system. The !-feature (see section 14) will permit you to edit and print the file without leaving MATLAB.

18. Graphics.

MATLAB can produce planar plots of curves, 3-D plots of curves, 3-D mesh surface plots, and 3-D faceted surface plots. The primary commands for these facilities are `plot`, `plot3`, `mesh`, and `surf`, respectively. An introduction to each of these is given below.

To preview some of these capabilities, enter the command `demo` and select some of the graphics options.

Planar plots.

The `plot` command creates linear x-y plots; if x and y are vectors of the same length, the command `plot(x,y)` opens a graphics window and draws an x-y plot of the elements of x versus the elements of y. You can, for example, draw the graph of the sine function over the interval -4 to 4 with the following commands:

```
x = -4:.01:4; y = sin(x); plot(x,y)
```

Try it. The vector x is a partition of the domain with meshsize 0.01 while y is a vector giving the values of sine at the nodes of this partition (recall that `sin` operates entrywise).

You will usually want to keep the current graphics window ("figure") exposed—but moved to the side—and the command window active.

One can have several graphics figures, one of which will at any time be the designated "current" figure where graphs from subsequent plotting commands will be placed. If, for example, figure 1 is the current figure, then the command `figure(2)` (or simply `figure`) will open a second figure (if necessary) and make it the current figure. The command `figure(1)` will then expose figure 1 and make it again the current figure. The command `gcf` will return the number of the current figure.

As a second example, you can draw the graph of $y = e^{-x^2}$ over the interval -1.5 to 1.5 as follows:

```
x = -1.5:.01:1.5; y = exp(-x.^2); plot(x,y)
```

Note that one must precede ^ by a period to ensure that it operates entrywise (see section 3).

MATLAB supplies a function `fplot` to easily and efficiently plot the graph of a function. For example, to plot the graph of the function above, one can first define the function in an M-file called, say, `expnormal.m` containing


```
function y = expnormal(x)
y = exp(-x.^2);
```

Then the command

```
fplot('expnormal', [-1.5,1.5])
```

will produce the graph. Try it.

Plots of parametrically defined curves can also be made. Try, for example,

```
t=0:.001:2*pi; x=cos(3*t); y=sin(2*t); plot(x,y)
```

The graphs can be given titles, axes labeled, and text placed within the graph with the following commands which take a string as an argument.

`title`	graph title
`xlabel`	x-axis label
`ylabel`	y-axis label
`gtext`	place text on the graph using the mouse
`text`	position text at specified coordinates

For example, the command

```
title('Best Least Squares Fit')
```

gives a graph a title. The command `gtext('The Spot')` allows one to interactively place the designated text on the current graph by placing the mouse pointer at the desired position and clicking the mouse. To place text in a graph at designated coordinates, one would use the command `text` (see `help text`).

The command `grid` will place grid lines on the current graph.

By default, the axes are auto-scaled. This can be overridden by the command `axis`. Some features of `axis` are:

`axis([`x_{min}`,`x_{max}`,`y_{min}`,`y_{max}`])`	set axis scaling to prescribed limits
`axis(axis)`	freezes scaling for subsequent graphs
`axis auto`	returns to auto-scaling
`v = axis`	returns vector v showing current scaling
`axis square`	same scale on both axes
`axis equal`	same scale and tic marks on both axes
`axis off`	turns off axis scaling and tic marks
`axis on`	turns on axis scaling and tic marks

The `axis` command should be given *after* the `plot` command.

Two ways to make multiple plots on a single graph are illustrated by

```
x=0:.01:2*pi;y1=sin(x);y2=sin(2*x);y3=sin(4*x);plot(x,y1,x,y2,x,y3)
```

and by forming a matrix `Y` containing the functional values as columns

```
x=0:.01:2*pi; Y=[sin(x)', sin(2*x)', sin(4*x)']; plot(x,Y)
```

Another way is with `hold`. The command `hold on` freezes the current graphics screen so that subsequent plots are superimposed on it. The axes may, however, become rescaled. Entering `hold off` releases the "hold."

One can override the default linetypes, pointtypes and colors. For example,

```
x=0:.01:2*pi; y1=sin(x); y2=sin(2*x); y3=sin(4*x);
plot(x,y1,'--',x,y2,':',x,y3,'+')
```

renders a dashed line and dotted line for the first two graphs while for the third the symbol `+` is placed at each node. The line- and mark-types are

Linetypes: solid (`-`), dashed (`--`). dotted (`:`), dashdot (`-.`)
Marktypes: point (`.`), plus (`+`), star (`*`), circle (`o`), x-mark (`x`)

Colors can be specified for the line- and mark-types.

Colors: yellow (`y`), magenta (`m`), cyan (`c`), red (`r`)
green (`g`), blue (`b`), white (`w`), black (`k`)

For example, `plot(x,y,'r--')` plots a red dashed line.

The command `subplot` can be used to partition the screen so that several small plots can be placed in one figure. See `help subplot`.

Other specialized 2-D plotting functions you may wish to explore via `help` are:

```
polar, bar, hist, quiver, compass, feather, rose, stairs, fill
```

Graphics hardcopy

A hardcopy of the current graphics figure can be most easily obtained with the MATLAB command `print`. Entered by itself, it will send a high-resolution copy of the current graphics figure to the default printer.

The `printopt` M-file is used to specify the default setting used by the `print` command. If desired, one can change the defaults by editing this file (see `help printopt`).

The command `print` *filename* saves the current graphics figure to the designated filename in the default file format. If *filename* has no extension, then an appropriate extension such as `.ps`, `.eps`, or `.jet` is appended. If, for example, PostScript is the default file format, then

```
print lissajous
```

will create a PostScript file `lissajous.ps` of the current graphics figure which can subsequently be printed using the system print command. If `filename` already exists, it will be overwritten unless you use the `-append` option. The command

```
print -append lissajous
```

will append the (hopefully different) current graphics figure to the existing file `lissajous.ps`. In this way one can save several graphics figures in a single file.

The default settings can, of course, be overwritten. For example,

```
print -deps -f3 saddle
```

will save to an Encapsulated PostScript file `saddle.eps` the graphics figure 3 — even if it is not the current figure.

3-D line plots.

Completely analogous to `plot` in two dimensions, the command `plot3` produces curves in three dimensional space. If x, y, and z are three vectors of the same size, then the command `plot3(x,y,z)` will produce a perspective plot of the piecewise linear curve in 3-space passing through the points whose coordinates are the respective elements of x, y, and z. These vectors are usually defined parametrically. For example,

```
t=.01:.01:20*pi; x=cos(t); y=sin(t); z=t.^3; plot3(x,y,z)
```

will produce a helix which is compressed near the x-y plane (a "slinky"). Try it.

Just as for planar plots, a title and axis labels (including `zlabel`) can be added. The features of `axis` command described there also hold for 3-D plots; setting the axis scaling to prescribed limits will, of course, now require a 6-vector.

3-D mesh and surface plots.

Three dimensional wire mesh surface plots are drawn with the command `mesh`. The command `mesh(z)` creates a three-dimensional perspective plot of the elements of the matrix z. The mesh surface is defined by the z-coordinates of points above a rectangular grid in the x-y plane. Try `mesh(eye(10))`.

Similarly, three dimensional faceted surface plots are drawn with the command `surf`. Try `surf(eye(10))`.

To draw the graph of a function $z = f(x, y)$ over a rectangle, one first defines vectors xx and yy which give partitions of the sides of the rectangle. With the function `meshgrid` one then creates a matrix x, each row of which equals xx and whose column length is the length of yy, and similarly a matrix y, each column of which equals yy, as follows:

```
[x,y] = meshgrid(xx,yy);
```

One then computes a matrix z, obtained by evaluating f entrywise over the matrices x and y, to which `mesh` or `surf` can be applied.

You can, for example, draw the graph of $z = e^{-x^2-y^2}$ over the square $[-2,2] \times [-2,2]$ as follows (try it):

```
xx = -2:.2:2;
yy = xx;
[x,y] = meshgrid(xx,yy);
z = exp(-x.^2 - y.^2);
mesh(z)
```

One could, of course, replace the first three lines of the preceding with

```
[x,y] = meshgrid(-2:.2:2, -2:.2:2);
```

Try this plot with `surf` instead of `mesh`.

As noted above, the features of the `axis` command described in the section on planar plots also hold for 3-D plots as do the commands for titles, axes labelling and the command `hold`.

The color shading of surfaces is set by the `shading` command. There are three settings for shading: `faceted` (default), `interpolated`, and `flat`. These are set by the commands

`shading faceted`, `shading interp`, or `shading flat`

Note that on surfaces produced by `surf`, the settings `interpolated` and `flat` remove the superimposed mesh lines. Experiment with various shadings on the surface produced above. The command `shading` (as well as `colormap` and `view` below) should be entered *after* the `surf` command.

The color profile of a surface is controlled by the `colormap` command. Available predefined colormaps include:

`hsv` (default), `hot`, `cool`, `jet`, `pink`, `copper`, `flag`, `gray`, `bone`

The command `colormap(cool)` will, for example, set a certain color profile for the current figure. Experiment with various colormaps on the surface produced above.

The command `view` can be used to specify in spherical or cartesian coordinates the viewpoint from which the 3-D object is to be viewed. See `help view`.

The MATLAB function `peaks` generates an interesting surface on which to experiment with `shading`, `colormap`, and `view`.

Plots of parametrically defined surfaces can also be made. The MATLAB functions `sphere` and `cylinder` will generate such plots of the named surfaces. (See `type sphere` and `type cylinder`.) The following is an example of a similar function which generates a plot of a torus.

```
function [x,y,z] = torus(r,n,a)
%TORUS Generate a torus
%   torus(r,n,a) generates a plot of a torus with central
%   radius a and lateral radius r.  n controls the number
%   of facets on the surface.  These input variables are optional
%   with defaults r = 0.5, n = 30, a = 1.
%
%   [x,y,z] = torus(r,n,a) generates three (n+1)-by-(n+1)
%   matrices so that surf(x,y,z) will produce the torus.
%
%   See also SPHERE, CYLINDER

if nargin < 3, a = 1; end
if nargin < 2, n = 30; end
if nargin < 1, r = 0.5; end
theta = pi*(0:2:2*n)/n;
phi = 2*pi*(0:2:n)'/n;
xx = (a + r*cos(phi))*cos(theta);
yy = (a + r*cos(phi))*sin(theta);
zz = r*sin(phi)*ones(size(theta));
if nargout == 0
   surf(xx,yy,zz)
   ar = (a + r)/sqrt(2);
   axis([-ar,ar,-ar,ar,-ar,ar])
else
```



```
   x = xx; y = yy; z = zz;
end
```

Other 3-D plotting functions you may wish to explore via `help` are:

`meshz, surfc, surfl, contour, pcolor`

Handle Graphics.

Beyond those described above, MATLAB's graphics system provides low level functions which permit one to control virtually all aspects of the graphics environment to produce sophisticated plots. Enter the command `set(1)` and `gca,set(ans)` to see some of the properties of figure 1 which one can control. This system is called Handle Graphics, for which one is referred to the MATLAB User's Guide.

19. Sparse Matrix Computations.

In performing matrix computations, MATLAB normally assumes that a matrix is dense; that is, any entry in a matrix *may* be nonzero. If, however, a matrix contains sufficiently many zero entries, computation time could be reduced by avoiding arithmetic operations on zero entries and less memory could be required by storing only the nonzero entries of the matrix. This increase in efficiency in time and storage can make feasible the solution of significantly larger problems than would otherwise be possible. MATLAB provides the capability to take advantage of the sparsity of matrices.

Matlab has two storage modes, full and sparse, with full the default. The functions `full` and `sparse` convert between the two modes. For a matrix A, full or sparse, `nnz(A)` returns the number of nonzero elements in A.

A sparse matrix is stored as a linear array of its nonzero elements along with their row and column indices. If a full tridiagonal matrix F is created via, say,

```
F = floor(10*rand(6)); F = triu(tril(F,1),-1);
```

then the statement `S = sparse(F)` will convert F to sparse mode. Try it. Note that the output lists the nonzero entries in column major order along with their row and column indices. The statement `F = full(S)` restores S to full storage mode. One can check the storage mode of a matrix A with the command `issparse(A)`.

A sparse matrix is, of course, usually generated directly rather than by applying the function `sparse` to a full matrix. A sparse banded matrix can be easily created via the function `spdiags` by specifying diagonals. For example, a familiar sparse tridiagonal matrix is created by

```
m = 6; n = 6; e = ones(n,1); d = -2*e;
T = spdiags([e,d,e],[-1,0,1],m,n)
```

Try it. The integral vector [-1,0,1] specifies in which diagonals the columns of [e,d,e] should be placed (use `full(T)` to view). Experiment with other values of m and n and, say, [-3,0,2] instead of [-1,0,1]. See `help spdiags` for further features of `spdiags`.

The sparse analogs of `eye`, `zeros`, `ones`, and `randn` for full matrices are, respectively,

```
speye,  sparse,  spones,  sprandn
```

The latter two take a matrix argument and replace only the nonzero entries with ones and normally distributed random numbers, respectively. `randn` also permits the sparsity structure to be randomized. The command `sparse(m,n)` creates a sparse zero matrix.

The versatile function `sparse` permits creation of a sparse matrix via listing its nonzero entries. Try, for example,

```
i = [1 2 3 4 4 4]; j = [1 2 3 1 2 3]; s = [5 6 7 8 9 10];
S = sparse(i,j,s,4,3), full(S)
```

In general, if the vector s lists the nonzero entries of S and the integral vectors i and j list their corresponding row and column indices, then

```
sparse(i,j,s,m,n)
```

will create the desired sparse $m \times n$ matrix S. As another example try

```
n = 6; e = floor(10*rand(n-1,1)); E = sparse(2:n,1:n-1,e,n,n)
```

The arithmetic operations and most MATLAB functions can be applied independent of storage mode. The storage mode of the result? Operations on full matrices always give full results. Selected other results are (S=sparse, F=full):

Sparse: `S+S, S*S, S.*S, S.*F, S^n, S.^n, S\S`
Full: `S+F, S*F, S\F, F\S`
Sparse: `inv(S), chol(S), lu(S), diag(S), max(S), sum(S)`

For sparse S, `eig(S)` is full if S is symmetric but undefined if S is unsymmetric; `svd` requires a full argument. A matrix built from blocks, such as `[A,B;C,D]`, is sparse if any constituent block is sparse.

You may wish to compare, for the two storage modes, the efficiency of solving a tridiagonal system of equations for, say, $n = 20, 50, 500, 1000$ by entering, recalling and editing the following two command lines:

```
n=20;e=ones(n,1);d=-2*e; T=spdiags([e,d,e],[-1,0,1],n,n); A=full(T);
b=ones(n,1);s=sparse(b);tic,T\s;sparsetime=toc, tic,A\b;fulltime=toc
```


20. Reference.

There are many MATLAB features which cannot be included in these introductory notes. Listed below are some of the MATLAB functions and operators available, grouped by subject area[1]. Use the on-line help facility or consult the Reference Guide for more detailed information on the functions.

There are many functions beyond these. There exist, in particular, several "toolboxes" of functions for specific areas[2]. Included among such are signal processing, control systems, robust-control, system identification, optimization, splines, chemometrics, μ-analysis and synthesis, state-space identification, neural networks, image processing, symbolic math (Maple kernel), and statistics. These can be explored via the command `help`.

Managing Commands and Functions	
help	help facility
what	list M-files on disk
type	list named M-file
lookfor	keywork search through the help entries
which	locate functions and files
demo	run demonstrations
path	control MATLAB's search path
cedit	set parameters for command line editing and recall
version	display MATLAB version you are running
whatsnew	display toolbox README files
info	info about MATLAB and The MathWorks
why	receive flippant answer

Managing Variables and the Workspace	
who	list current variables
whos	list current variables, long form
save	save workspace variables to disk
load	retrieve variables from disk
clear	clear variables and functions from memory
pack	consolidate workspace memory
size	size of matrix
length	length of vector
disp	display matrix or text

[1] Source: MATLAB Reference Guide, version 4.1

[2] The toolboxes, which are optional, may not be installed on your system.

Working with Files and the Operating System	
cd	change current working directory
pwd	show current working directory
dir, ls	directory listing
delete	delete file
getenv	get environment variable
!	execute operating system command
unix	execute operating system command; return result
diary	save text of MATLAB session

Controlling the Command Window	
clc	clear command window
home	send cursor home—to top of screen
format	set output format
echo	echo commands inside script commands
more	control paged output in command window

Starting and Quitting from MATLAB	
quit	terminate MATLAB
startup	M-file executed when MATLAB is started
matlabrc	master startup M-file

Matrix Operators		Array Operators	
+	addition	+	addition
−	subtraction	−	subtraction
*	multiplication	.*	multiplication
^	power	.^	power
/	right division	./	right division
\	left division	.\	left division
'	conjugate transpose		
.'	transpose		
kron	Kronecker tensor product		

Relational and Logical Operators			
<	less than	&	and
<=	less than or equal	|	or
>	greater than	∼	not
>=	greater than or equal	**xor**	exclusive or
==	equal		
∼=	not equal		

Special Characters	
=	assignment statement
[ ]	used to form vectors and matrices; enclose multiple function output variables
( )	arithmetic expression precedence; enclose function input variables
.	decimal point
..	parent directory
...	continue statement to next line
,	separate subscripts, function arguments, statements
;	end rows, suppress printing
%	comments
:	subscripting, vector generation
!	execute operating system command

Special Variables and Constraints	
ans	answer when expression not assigned
eps	floating point precision
realmax	largest floating point number
reammin	smallest positive floating point number
pi	π
i, j	imaginary unit
inf	infinity
NaN	Not-a-Number
flops	floating point operation count
nargin	number of function input arguments
nargout	number of function output arguments
computer	computer type

Time and Date	
date	current date
clock	wall clock
etime	elapsed time function
tic, toc	stopwatch timer functions
cputime	elapsed CPU time

Special Matrices	
zeros	matrix of zeros
ones	matrix of ones
eye	identity
diag	diagonal
toeplitz	Toeplitz
magic	magic square
compan	companion
linspace	linearly spaced vectors
logspace	logarithmically spaced vectors
meshgrid	array for 3-D plots
rand	uniformly distributed random numbers
randn	normally distributed randon numbers
hilb	Hilbert
invhilb	inverse Hilbert (exact)
vander	Vandermonde
pascal	Pascal
hadamard	Hadamard
hankel	Hankel
rosser	symmetric eigenvalue test matrix
wilkinson	Wilkinson's eigenvalue test matrix
gallery	two small test matrices

Matrix Manipulation	
diag	create or extract diagonals
rot90	rotate matrix 90 degrees
fliplr	flip matrix left-to-right
flipud	flip matrix up-to-down
reshape	change size
tril	lower triangular part
triu	upper triangular part
.'	transpose
:	convert matrix to single column; `A(:)`

Logical Functions	
exist	check if variables or functions exist
any	true if any element of vector is true
all	true if all elements of vector are true
find	find indices of non-zero elements
isnan	true for NaNs
isinf	true for infinite elements
finite	true for finite elements
isieee	true for IEEE floating point arithmetic
isempty	true for empty matrix
issparse	true for sparse matrix
isstr	true for text string
strcmp	compare string variables

Control Flow	
if	conditionally execute statements
else	used with **if**
elseif	used with **if**
end	terminate **if, for, while**
for	repeat statements for a specific number of times
while	repeat statments while condition is true
break	terminate execution of **for** or **while** loops
return	return to invoking function
error	display message and abort function

Programming	
input	prompt for user input
keyboard	invoke keyboard as if it were a script file
menu	generate menu of choices for user input
pause	wait for user response
function	define function
eval	execute string with MATLAB expression
feval	evaluate function specified by string
global	define global variables
nargchk	validate number of input arguments

Text and Strings	
string	about character strings in MATLAB
abs	convert string to numeric values
blanks	a string of blanks
eval	evaluate string with MATLAB expression
num2str	convert number to string
int2str	convert integer to string
str2num	convert string to number
isstr	true for string variables
strcmp	compare string variables
upper	convert string to uppercase
lower	convert string to lowercase
hex2num	convert hex string to floating point number
hex2dec	convert hex string to decimal integer
dec2hex	convert decimal integer to hex string

Debugging	
dbstop	set breakpoint
dbclear	remove breakpoint
dbcont	remove execution
dbdown	change local workspace context
dbstack	list who called whom
dbstatus	list all breakpoints
dbstep	execute one or more lines
dbtype	list M-file with line numbers
dbup	change local workspace context
dbdown	opposite of **dbup**
dbquit	quit debug mode

Sound Processing Functions	
saxis	sound axis scaling
sound	convert vector to sound
auread	Read Sun audio file
auwrite	Write Sun audio file
lin2mu	linear to mu-law conversion
mu2lin	mu-law to linear conversion

Elementary Math Functions	
abs	absolute value or complex magnitude
angle	phase angle
sqrt	square root
real	real part
imag	imaginary part
conj	complex conjugate
gcd	greatest common divisor
lcm	least common multiple
round	round to nearest integer
fix	round toward zero
floor	round toward $-\infty$
ceil	round toward ∞
sign	signum function
rem	remainder
exp	exponential base e
log	natural logarithm
log10	log base 10

Trigonometric Functions	
sin, asin, sinh, asinh	sine, arcsine, hyperbolic sine, hyperbolic arcsine
cos, acos, cosh, acosh	cosine, arccosine, hyperbolic cosine, hyperbolic arccosine
tan, atan, tanh, atanh	tangent, arctangent, hyperbolic tangent, hyperbolic arctangent
cot, acot, coth, acoth	cotangent, arccotangent, hyperbolic cotan., hyperbolic arccotan.
sec, asec, sech, asech	secant, arcsecant, hyperbolic secant, hyperbolic arcsecant
csc, acsc, csch, acsch	cosecant, arccosecant, hyperbolic cosecant, hyperbolic arccosecant

Special Functions	
bessel	bessel function
beta	beta function
gamma	gamma function
rat	rational approximation
rats	rational output
erf	error function
erfinv	inverse error function
ellipke	complete elliptic integral
ellipj	Jacobian elliptic integral
expint	exponential integral
log2	dissect floating point numbers
pow2	scale floating point numbers

Matrix Decompositions and Factorizations	
inv	inverse
lu	factors from Gaussian elimination
rref	reduced row echelon form
chol	Cholesky factorization
qr	orthogonal-triangular decomposition
nnls	nonnegative least squares
lscov	least squares in presence of know covariance
null	null space
orth	orthogonalization
eig	eigenvalues and eigenvectors
hess	Hessenberg form
schur	Schur decomposition
cdf2rdf	complex diagonal form to real block diagonal form
rsf2csf	real block diagonal form to complex diagonal form
balance	diagonal scaling for eigenvalue accuracy
qz	generalized eigenvalues
polyeig	polynomial eigenvalue solver
svd	singular value decomposition
pinv	pseudoinverse

Matrix Conditioning	
cond	condition number in 2-norm
rcond	LINPACK reciprocal condition number estimator
condest	Hager/Higham condition number estimator
norm	1-norm,2-norm,F-norm,∞-norm
normest	2-norm estimator
rank	rank

Elementary Matrix Functions	
expm	matrix exponential
expm1	M-file implementation of **expm**
expm2	matrix exponential via Taylor series
expm3	matrix exponential via eigenvalues and eigenvectors
logm	matrix logarithm
sqrtm	matrix square root
funm	evaluate general matrix function
poly	characteristic polynomial
det	determinant
trace	trace

Polynomials	
poly	construct polynomial with specified roots
roots	polynomial roots—companion matrix method
roots1	polynomial roots—Laguerre's method
polyval	evaluate polynomial
polyvalm	evaluate polynomial with matrix argument
conv	multiply polynomials
deconv	divide polynomials
residue	partial-fraction expansion (residues)
polyfit	fit polynomial to data
polyder	differentiate polynomial

Column-wise Data Analysis	
max	largest component
min	smallest component
mean	average or mean value
median	median value
std	standard deviation
sort	sort in ascending order
sum	sum of elements
prod	product of elements
cumsum	cumulative sum of elements
cumprod	cumulative product of elements
hist	histogram

Signal Processing	
abs	complex magnitude
angle	phase angle
conv	convolution and polynomial multiplication
deconv	deconvolution and polynomial division
corrcoef	correlation coefficients
cov	covariance matrix
filter	one-dimensional digital filter
filter2	two-dimensional digital filter
cplxpair	sort numbers into complex pairs
unwrap	remove phase angle jumps across 360° boundaries
nextpow2	next higher power of 2
fft	radix-2 fast Fourier transform
fft2	two-dimensional FFT
ifft	inverse fast Fourier transform
ifft2	inverse 2-D FFT
fftshift	zero-th lag to center of spectrum

Finite Differences and Data Interpolation	
diff	approximate derivatives
gradient	approximate gradient
del2	five point discrete Laplacian
subspace	angle between two subspaces
spline	cubic spline interpolation
interp1	1-D data interpolation
interp2	2-D data interpolation
interpft	1-D data interpolation via FFT method
griddata	data gridding

Numerical Integration	
quad	adaptive 2-panel Simpson's Rule
quad8	adaptive 8-panel Newton-Cotes Rule
trapz	trapezoidal method

Differential Equation Solution	
ode23	2nd/3rd order Runge-Kutta method
ode23p	solve via **ode23**, displaying plot
ode45	4th/5th order Runge-Kutta-Fehlberg method

Nonlinear Equations and Optimization	
fmin	minimize function of one variable
fmins	minimize function of several variables
fsolve	solution to a system of nonlinear equations (find zeros of a function of several variables)
fzero	find zero of function of one variable
fplot	plot graph of a function

Two Dimensional Graphs	
plot	linear plot
loglog	log-log scale plot
semilogx	semilog scale plot
semilogy	semilog scale plot
fill	draw filled 2-D polygons
polar	polar coordinate plot
bar	bar graph
stairs	stairstep plot
errorbar	error bar plot
hist	histogram plot
rose	angle histogram plot
compass	compass plot
feather	feather plot
fplot	plot function

Graph Annotation	
title	graph title
xlabel	x-axis label
ylabel	y-axis label
zlabel	z-axis label for 3-D plots
grid	grid lines
text	text annotation
gtext	mouse placement of text
ginput	graphical input from mouse

Figure Window/Axis Creation and Control	
figure	create figure (graph window)
gcf	get handle to current figure
clf	clear current figure
close	close figure
hold	hold current graph
ishold	return hold status
subplot	create axes in tiled positions
axes	create axes in arbitrary positions
gca	get handle to to current axes
axis	control axis scaling and appearance
caxis	control pseudocolor axis scaling

Graph Hardcopy and Storage	
print	print graph or save graph to file
printopt	configure local printer defaults
orient	set paper orientation

Three Dimensional Graphs	
mesh	3-D mesh surface
meshc	combination mesh/contour plot
meshz	3-D mesh with zero plane
surf	3-D shaded surface
surfc	combination surface/contour plot
surfl	3-D shaded surface with lighting
plot3	plot lines and points in 3-D space
fill3	draw filled 3-D polygons in 3-D space
contour	contour plot
contour3	3-D contour plot
clabel	contour plot elevation labels
contourc	contour plot computation (used by `contour`)
pcolor	pseudocolor (checkerboard) plot
quiver	quiver plot
image	display image
waterfall	waterfall plot
slice	volumetric visualization plot

3-D Graph Appearance	
view	3-D graph viewpoint specification
viewmtx	view transformation matrices
hidden	mesh hidden line removal mode
shading	color shading mode
axis	axis scaling and apearance
caxis	pseudocolor axis scaling
specular	specular reflectance
diffuse	diffuse reflectance
surfnorm	surface normals
colormap	color lookup table (see below)
brighten	brighten or darken color map
spinmap	spin color map
rgbplot	plot colormap
hsv2rgb	hsv to rgb color map conversion
rgb2hsv	rgb to hsv color map conversion

Color Maps	
hsv	hue-saturation-value (default)
jet	variant of `hsv`
gray	linear gray-scale
hot	black-red-yellow-white
cool	shades of cyan and magenta
bone	gray-scale with tinge of blue
copper	linear copper tone
pink	pastel shades of pink
flag	alternating red, white, blue, and black

3-D Objects	
sphere	generate sphere
cylinder	generate cylinder
peaks	generate demo surface

Movies and Animation	
moviein	initialize movie frame memory
getframe	get movie frame
movie	play recorded movie frames

Handle Graphics Objects	
figure	create figure window
axes	create axes
line	create line
text	create text
patch	create patch
surface	create surface
image	create image
uicontrol	create user interface control
uimenu	create user interface menu

Handle Graphics Operations	
set	set object properties
get	get object properties
reset	reset object properties
delete	delete object
drawnow	flush pending graphics events

Sparse Matrix Functions	
spdiags	sparse matrix formed from diagonals
speye	sparse identity matrix
sprandn	sparse random matrix
spones	replace nonzero entries with ones
sprandsym	sparse symmetric random matrix
spfun	apply function to nonzero entries
sparse	create sparse matrix; convert full matrix to sparse
full	convert sparse matrix to full matrix
find	find indices of nonzero entries
spconvert	convert from sparse matrix external format
issparse	true if matrix is sparse
nnz	number of nonzero entries
nonzeros	nonzero entries
nzmax	amount of storage allocated for nonzero entries
spalloc	allocate memory for nonzero entries
spy	visualize sparsity structure
gplot	plot graph, as in "graph theory"
colmmd	column minimum degree
colperm	order columns based on nonzero count
dmperm	Dulmage-Mendelsohn decomposition
randperm	random permutation vector
symmmd	symmetric minimum degree
symrcm	reverse Cuthill-McKee ordering
condest	estimate 1-norm condition
normest	estimate 2-norm
sprank	structural rank
spaugment	form least squares augmented system
spparms	set parameters for sparse matrix routines
symbfact	symbolic factorization analysis
sparsefun	sparse auxillary functions and parameters

B Travaux pratiques d'analyse numérique avec MATLAB

B.1 Introduction

Vous disposez de 5 séances pour réaliser les 4 travaux pratiques ci-dessous. Les TP 1 et 3 doivent s'effectuer en 2 séances environ, le TP 2 ne doit nécessiter qu'une séance.

B.2 Enoncés des TP

B.2.1 Travaux pratiques 1: Approximation différences-finies des équations elliptiques et conditions aux Limites

Ce TP concerne la discrétisation différences-finies de problèmes elliptiques 1D, de type convection-diffusion stationnaire ou Helmholtz déjà abordés dans les exercices A.8 et A.9. Le répertoire de travail MATLAB du TP est *'TP MATLAB CONV DIFF STATIONNAIRE'*.

B.2.2 Equation de convection-diffusion stationnaire

Cette partie concerne le problème aux limites (A.1) de l'exercice A.8.

Le programme `Tp_matlabconv_diff_stationnaire.m` résoud cette EDP pour $a = 1$ et $\nu = 0.02$ (variables MATLAB `adv` et `nu` respectivement) par méthode de différences finies centrées (fonction `schema_centre_convdiff_stat.m`). Ensuite, il trace sur le même graphe la solution numérique (vecteur `unum`) et la solution exacte (définie dans la fonction `uex_convdiff`) aux points du maillage.

Question 1. *Exécutez le programme pour $n = 11$ points et calculez le nombre de Péclet de maille Pe_h. Que remarquez-vous ? Calculez le nombre minimal de points pour que les problèmes numériques disparaissent. Exportez les graphes nécessaires pour le compte-rendu final.*

On veut maintenant vérifier numériqument que la méthode de différences finies centrées est au second-ordre, en calculant pour plusieurs valeurs du pas d'espace h la norme l^2 de l'erreur :

$$\|u_{\text{ex}} - u_h\|_{l^2} \tag{B.1}$$

où $\|\cdot\|_{l^2}$ représente l'erreur quadratique moyenne aux points intérieurs du maillage, définie par

$$\|\phi\|_{l^2}^2 = \frac{1}{N}\sum_{i=1}^{N} \phi(x_i)^2. \tag{B.2}$$

Question 2. *Tracez sur un graphe en coordonnées logarithmiques (commande `loglog`) l'erreur l^2 de la méthode de différences-finies centrées obtenue pour des valeurs croissantes[1] de n, et montrez qu'elle est au second-ordre en traçant sur le même graphe la fonction $n \mapsto 1/n^2$.*

Question 3. *Programmez la méthode de différences finies upwind. Par expérimentation numérique, vérifiez que la solution numérique ne présente pas d'oscillations parasites quelle que soit la valeur de Pe_h.*

[1] Conseil : prenez des valuers de n supérieures à 50.

B.2.3 Equation de Helmholtz et conditions aux limites de Neumann

Le but de cette partie est d'établir numériquement l'ordre de précision de schémas aux différences finies centrés pour le problème de Helmholtz avec les conditions aux limtes de type Neumann-Dirichlet :

$$\lambda u - \frac{\mathrm{d}^2 u}{\mathrm{d}x^2} = f(x), \qquad \lambda \geq 0, \tag{B.3a}$$

$$u'(-1) = \sqrt{\lambda}\tanh^{-1}(\sqrt{\lambda}), \qquad u(1) = 1, \tag{B.3b}$$

dans $\Omega =]-1, 1[$, en utilisant différentes manières d'imposer une condition aux limites de Neumann, telle que cela a été vu dans l'exercice A.9.

Solution exacte du problème de Helmholtz

Pour $f(x) = \lambda$, la solution exacte de (B.3) s'écrit

$$u_\lambda(x) = 1 - \frac{\sinh\sqrt{\lambda}\,(1-x)}{\sinh 2\sqrt{\lambda}}. \tag{B.4}$$

On peut observer que $u_\lambda(x) \sim 1$ lorsque $\lambda \gg 1$ sauf dans un voisinage du bord $x = -1$, de taille $\mathcal{O}(1/\sqrt{\lambda})$, où la solution exhibe une couche limite.

Implémentation d'une condition de Neumann

On cherche à évaluer les estimations d'erreur (B.1)pour 3 types d'implémentations d'une C.L. de Neumann :

I) Méthode de l'exemple 7.3 du cours,

II) Méthode de l'exemple 7.4,

III) Méthode vue en *TD*.

Qui va l'emporter ?

Question 4. *Placez-vous dans le cas $\lambda = 100$. Programmez la méthode de différences-finies centrées avec les trois variantes de C.L. (faites une fonction MATLAB pour chaque méthode).*

Question 5. *Comparez les estimations d'erreur obtenue pour chaque méthode. Sont-elles consistantes avec la théorie ? Distinguez la meilleure de ces trois méthodes.*

Question 6. *En utilisant les graphes de la question précédente, estimez le nombre de nœuds nécessaires pour que le schéma du premier ordre fournisse une précision (en norme l^2) comparable à celle donnée par la méthode III avec $n = 100$ points.*

B.3 Travaux pratiques 2 : Schémas Numériques pour les Equations Hyperboliques

B.3.1 Introduction

On s'intéresse à la résolution numérique de l'équation de transport :

$$\frac{\partial u}{\partial t} + a\,\frac{\partial u}{\partial x} = 0 \tag{B.1a}$$

dans le domaine $\Omega =]a_x, b_x[\times]0, t_\mathrm{f}[$, avec la condition initiale

$$u(x, t = 0) = u_0(x), \tag{B.1b}$$

et la condition d'*inflow*

$$u(a_x, t) = g_-(t). \tag{B.1c}$$

Le répertoire de travail MATLABdu TP est *'TP MATLAB EQT HYPERB'*.

B.3.2 Condition initiale régulière

Dans cette première partie, on s'intéresse aux solutions de (B.1) qui restent continues en tout instant.

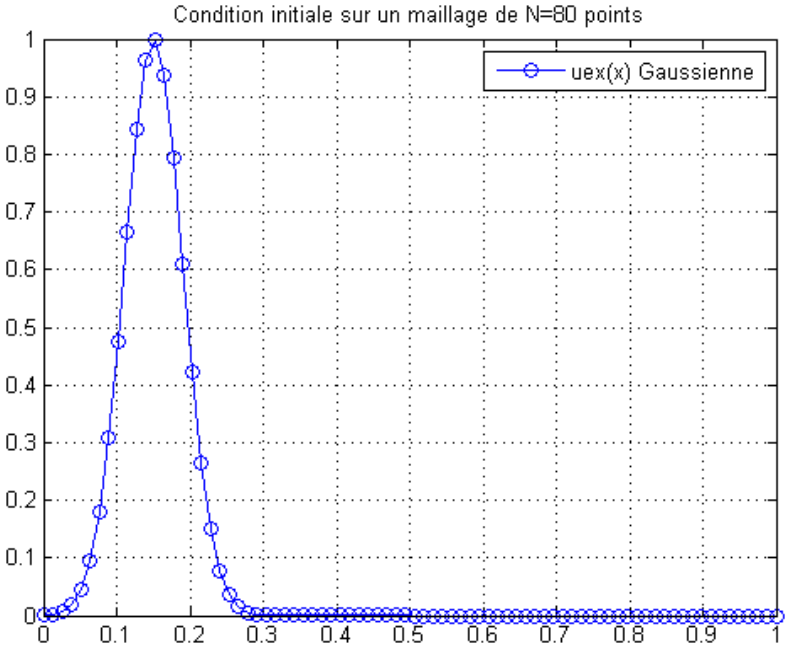


Fig. B.1. Condition initiale (B.2) avec $\sigma = 0.04$ et $x_0 = 0.15$.

.

Familiarisation avec le problème

On choisi le domaine $\Omega =]0, 1[$, et le coefficient d'advection de l'E.D.P. est $a = 1$ (variable "adv" dans le code MATLAB). La condition initiale est définie comme

$$u_0(x) = \exp\left(\frac{1}{2}\left[\frac{x - x_0}{\sigma}\right]^2\right), \tag{B.2}$$

qui correspond à une gaussienne d'écart type σ, centrée en $x_0 \in \Omega$ (*cf.* figure B.1). Plus la valeur de σ est grande, plus le problème est raide. On rappelle que la solution du problème (B.1) correspond à une translation uniforme de la condition initiale

Remarque B.1. Le problème (B.1) étant instationnaire, il faut fixer la borne supérieure d'intégration en temps T. Dans le code MATLAB, on définira

$$t_{\mathrm{f}} \approx \frac{b_x - a_x}{a},$$

de façon que l'information toute entière soit sortie du domaine Ω.

L'approximation numérique

On va comparer qualitativement divers schémas pour la résolution de (B.1). Les schémas considérés sont les suivants :

- Schéma upwind (*cf.* cours),
- $\theta-$schéma (*cf.* TD), pour $\theta \in [0,1]$:

$$\frac{u_i^{n+1}-u_i^n}{\Delta t} + a\left[\theta\,\delta_x^0 u_i^{n+1} + (1-\theta)\,\delta_x^0 u_i^n\right] = 0, \tag{B.3}$$

- Schéma de Lax-Wendroff :

$$\frac{u_i^{n+1}-u_i^n}{\Delta t} + a\,\delta_x^0 u_i^n - a^2\frac{\Delta t}{2}\delta_{xx}^0 u_i^n = 0. \tag{B.4}$$

Le schéma upwind est déjà programmé (fonction `unew=schema_upwind(c,uold,ug)`). Ces schémas sont exprimés en fonction du nombre de courant C uniquement (variable "cfl" dans le code MATLAB). On rappelle la définition de C :

$$C = \frac{a\Delta t}{\Delta x}. \tag{B.5}$$

Remarque B.2. Le problème (B.1) est bien posé avec l'unique condition aux limites amont (B.1c). Il est à noter que les schémas centrés, en particulier le $\theta-$schéma (B.3) et le schéma de Lax-Wendroff (B.4) nécessitent d'imposer une condition de sortie numérique en $x = b_x$, qui est donc artificielle.

La comparaison des schémas

Le but de cette étude est d'observer par la pratique les notions qui ont été vues "théoriquement" en cours et en TD, *c.à.d* :

- Stabilité,
- Dissipation + dispersion numérique,
- Oscillations parasites.

Pour chacun des schémas, il s'agira d'évaluer qualitativement les propriétés énoncées ci-dessus et d'en effectuer une synthèse. A la différence du *TP* précédent, nous ne ferons pas de raffinement de maillage. On conseille ici de conserver $N = 80$ points. Il est conseillé de prendre des notes manuscrites lors de l'évaluation de chacun des schémas.

Question 1. *Le progamme MATLABTp_matlab_eqt_hyperboliques.m utilise le schéma upwind pour résoudre l'équation de transport pour la condition initiale* (B.2) *avec* $\sigma = 0.04$ *et* $x_0 = 0.15$. *Utilisez ce schéma avec* $C = 1.2$. *Que se passe-t-il ? Comment remédier au problème ? Observez et évaluez les phénomènes de dissipation et dispersion numérique.*

Remarque B.3. Pour des valeurs de Δx et a données, il est équivalent de fixer la valeur de Δt ou celle de C, d'après la définition (B.5).

Désormais, la comparaison des différents schémas sera effectuée pour une même valeur du CFL.

Question 2. *Programmez le $\theta-$schéma de la manière qui a été définie en TD.*

Question 3. *Utilisez le $\theta-$schéma avec $\theta = 0$. Donnez le nom de ce schéma ? Quels sont les résultats obtenus ? Pouviez-vous le prévoir ?*

Question 4. *$\theta-$schéma avec $\theta = 1/2$. Le reconnaissez-vous ? Quel est son ordre de précision ? Utilisez le avec $C = 0.8$, 1.2, 2, etc ... Quelles sont ses propriétés de stabilité ? Pouviez-vous les prévoir ? Observez et évaluez les phénomènes de dissipation et dispersion numériques.*

Question 5. *$\theta-$schéma avec $\theta = 1$. Même questions que précédemment.*

Question 6. *Programmez le schéma de Lax-Wendroff. Rappelez l'ordre de précision de la méthode ? Retrouve-t'on les propriétés de stabilité théoriques ? Quelles sont les principales caractéristiques de la solution numérique par rapport à celle donnée par le schéma upwind ?*

Question 7. *Synthèse des résultats : parmi les schémas que vous venez d'utiliser, énoncez selon-vous les trois meilleurs et justifiez brièvement pourquoi vous rejetez les autres. Pour les trois meilleurs schémas, rappelez leurs propriétés et avantages et énoncez un classement subjectif selon ce que vous avez observé.*

B.3.3 Condition initiale discontinue

Les problèmes de mécanique des fluides compressibles, modélisés par des équations hyperboliques, admettent des chocs, *i.e.* des discontinuités dans la solution. Comme prototype (linéaire), nous allons considérer le problème (B.1) avec la condition initiale de Riemann :

$$u_0(x) = \begin{cases} 1 \text{ si } x < x_0, \\ \\ 0 \text{ si } x > x_0, \end{cases} \tag{B.6}$$

qui va se propager à la vitesse $a > 0$.

Il est, bien sûr, impossible d'obtenir une grande précision numérique au travers d'un choc. Il est néanmoins important de développer des méthodes numériques pour la mécanique des fluides telles que :

- une grande précision est obtenue loin du choc,
- il y a peu d'oscillations parasites près des discontinuités,
- La solution numérique conserve un profil de choc nettement défini, et non atténué par une trop forte viscosité artificielle.

Question 8. *Comparez les résultats des meilleurs schémas de la section B.3.2 lorsqu'ils sont appliqués à ce problème. Parmi les propriétés enoncées ci-dessus, lesquelles sont observées ? Conclusion + classement.*

B.4 Travaux pratiques 3 : Modélisation du transport d'un polluant par la méthode de volumes finis

Ce TP concerne la modélisation d'un polluant dispersé par un fluide tel celui représenté sur la figure B.2 (gauche). Le transport de ce polluant est modélisé par une équation de convection-diffusion 2D qui sera discrétisée par la méthode des volumes finis. Le répertoire de travail MATLAB du TP est *'TP MATLAB 2D CONV DIFF'*.

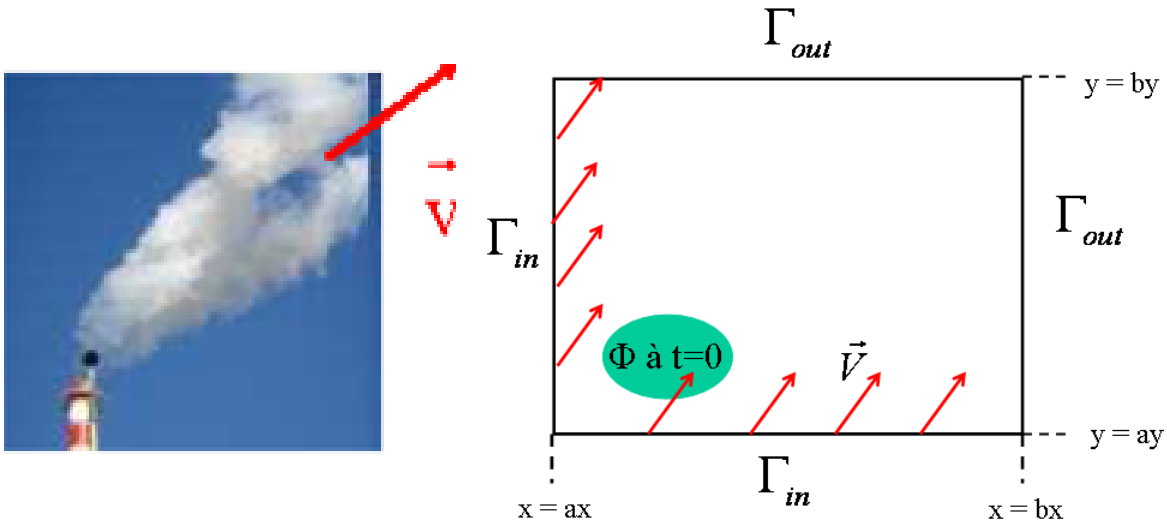


Fig. B.2. A gauche : dispersion de fumée dans l'atmosphère. A droite : Domaine de calcul considéré.

L'objectif du TP est de modéliser la dispersion de polluant (de fraction massique $\phi(x, y, t)$) à la surface d'un domaine fluide représenté par le domaine de calcul $\Omega =]a_x, b_x[\times]a_y, b_y[$. L'équation d'équilibre pour ϕ traduit que la variation temporelle de la quantité de polluant $\int_\Omega \rho\phi \, \mathrm{d}V$ dans un volume élémentaire Ω (de volume V, de surface Γ, et de normale extérieure $\boldsymbol{n}$) est égale à un bilan de flux $\int_\Gamma \boldsymbol{F}\cdot\boldsymbol{n} \, \mathrm{d}\Gamma$ sur les faces Γ du volume élémentaire. Ce flux est modélisé par la somme d'un flux de diffusion :

$$\boldsymbol{F}_\mathrm{d} = \lambda\boldsymbol{\nabla}\phi, \qquad \text{où } \lambda > 0 \text{ est le coefficient de diffusion,} \tag{B.1}$$

et d'un flux de convection :

$$\boldsymbol{F}_\mathrm{c} = -\rho\boldsymbol{v}\phi, \qquad \text{tel que } \boldsymbol{v} \text{ est la vitesse du fluide.} \tag{B.2}$$

On suppose que la vitesse du fluide est telle que représentée sur la figure B.2, et on distinguera les frontières d'*inflow* Γ_in (telles que $\boldsymbol{v}\cdot\boldsymbol{n} < 0$) où sont imposées des conditions de Dirichlet homogènes ($\phi_{|\Gamma_\mathrm{in}} = 0$), et les frontières d'*outflow* Γ_out (telles que $\boldsymbol{v}\cdot\boldsymbol{n} > 0$) où le fluide transporte le polluant à l'extérieur du domaine de calcul, et où sont imposées des conditions de Neumann homogènes ($\frac{\partial\phi}{\partial\boldsymbol{n}}_{|\Gamma_\mathrm{out}} = 0$).

A l'instant initial ($t = 0$), le polluant est placé à la position (x_0, y_0) du domaine de calcul et il est modélisé par :

$$\phi_0(x, y) = \mathrm{e}^{-(\frac{x-x_0}{\sigma_0})^2}\,\mathrm{e}^{-(\frac{y-y_0}{\sigma_0})^2} \tag{B.3}$$

qui correspond à la une gaussienne d'amplitude 1 et de rayon σ_0 représentée sur la figure B.3.

Le domaine de calcul $\Omega =]a_x, b_x[\times]a_y, b_y[$ est représenté par un maillage uniforme centré en cellules ($\Omega_h = \{\Omega_{i,j}, i = 2, \ldots, N_x, \, j = 2, \ldots, N_y\}$), de centre $\boldsymbol{x}_c(i, j) = (x_c(i), y_c(j))$, de volume $V_{i,j} = \Delta x\Delta y$. Pour imposer les conditions aux limites, des cellules fantômes sont créées sur les parois West $\{\Omega_{1,j}, \, j = 2, \ldots N_y\}$), East ($\{\Omega_{N_x+1,j}, \, j = 2, \ldots N_y\}$), South ($\{\Omega_{i,1}, \, i = 2, \ldots N_x\}$) et North ($\{\Omega_{i,N_y+1}, \, i = 2, \ldots N_x\}$).

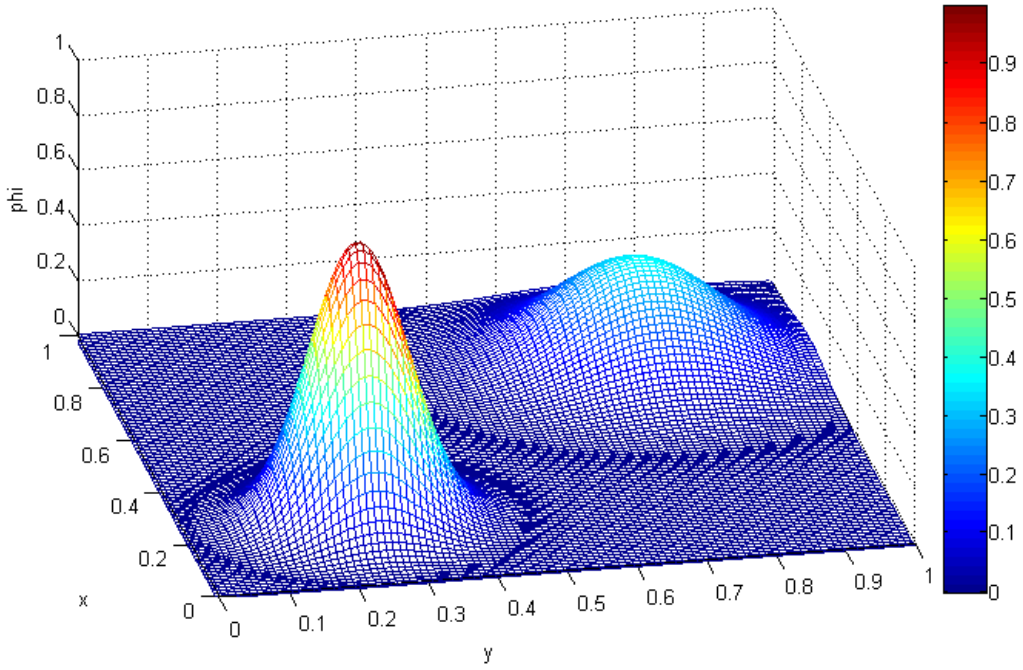


Fig. B.3. Solution initiale et finale du transport d'une gaussienne placée en $(x_0, y_0) = (1, 1))$ et de rayon $\sigma_0 = 0.1$.

Question 1. *Ecrivez la forme conservative de l'équation d'équilibre de ϕ dans une cellule $\Omega_{i,j}$ du maillage, en notant $\eta = \lambda/\rho$ le paramètre de diffusion.*

Question 2. *Faites un schéma du maillage complet en positionnant les variables discrètes $\{\phi_{i,j}\}$. Donnez la valeur que prend $\phi_{i,j}$ dans les cellules fantômes seolon le type de conditions aux limites.*

Question 3. *Effectuez une semi-discrétisation de l'équation de convection-diffusion pour ϕ par méthode de volumes-finis centrés en espace. Achevez-la en utilisant le schéma d'Euler progressif (EP1). Ecrivez le schéma complet sous sa forme itérative.*

Le programme MATLAB `'Tp_matlabconv_diff_2D.m'` crée le maillage Ω_h, discrétise la condition initiale (B.3) (écrite dans le programme `'phi_init.m'`) aux noeuds $\boldsymbol{x}_c(i,j)$ du maillage, et la représente sur un graphe (fonction `mesh(X,Y,Z)`). Vous allez le compléter en écrivant la boucle en temps du schéma EP1 centré et réaliser une simulation complète de transport du polluant.

Question 4. *Ecrivez la fonction MATLAB*

```
phinew=schema_EP1_centre(advx,advy,eta,dt,dx,dy,ncvx,ncvy,phiold)
```

qui correspond à une itération en temps du schéma d'Euler progressif, tel que :

- *les réels* `advx` *et* `advy` *correspondent aux composantes cartésiennes de la vitesse d'advection $\boldsymbol{v}$, supposée constante.*
- *Le tableau* `phiold` *de taille $(ncvx+1) \times (ncvy+1)$ doit contenir la solution au temps t_n avec conditions aux limites dans les cellules fantômes.*
- *Le tableau* `phinew` *de taille $(ncvx+1)\times(ncvy+1)$ donnera la solution au temps $t_{n+1} = (n+1)\Delta t$.*

La simulation que vous effectuerez prendra les paramètres suivants pour le domaine spatial :

$$a_x = 0, \qquad b_x = 1, \qquad a_y = 0, \qquad b_y = 1.$$

La condition initiale prend les paramètres suivants :

$$\sigma = 0.1, \qquad (x_0, y_0) = (1/4, 1/4),$$

et les paramètres de l'équation de convection diffusion sont :

$$\eta = 0.01, \qquad \boldsymbol{v} = (1, 1).$$

Question 5. *Cette question va permettre de fixer les paramètres de l'intégration en temps.*

- *Choisissez un intervalle de temps $[0, t_f]$ telle que le polluant soit sorti du domaine à $t = t_f$.*
- *Etablissez les conditions de stabilité du schéma EP1 centré (cf. cours) pour les paramêtres de la simulation.*

Question 6. *Programmez dans* `'Tp_matlabconv_diff_2D.m'` *la boucle d'intégration en temps du schéma EP1 centré (conseil : inspirez vous de la boucle* `while ... end` *du TP2). Vous serez amené à faire une animation de la solution numérique (commandes* `getframe` *et* `movie` *du TP2). Des indications sur l'écriture de la boucle en temps sont données dans le programme MATLAB.*

Question 7. *Effectuez des simulations pour une valeur de Δt supérieure et inférieure à la condition de stabilité. Qu'observez-vous ? Dans le compte-rendu, montrez le graphe de la solution à des temps bien choisis, et rendez le lsting de vos programmes MATLAB.*

Lab Exercice on Computational Fluid Dynamics

First steps with ANSYS FLUENT: The axysimmetric expansion flow

Instructor: Olivier Botella olivier.botella@univ-lorraine.fr

1 Problem statement

The purpose of this lab exercise[1] is to learn the basics of CFD analysis with the commercial software ANSYS FLUENT[2] by computing the axisymmetric expansion flow presented in the lectures (Fig. 1).

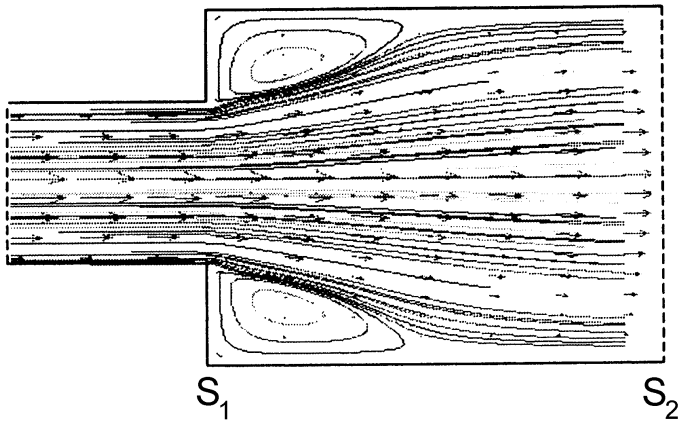


Figure 1: Velocity vectors at the vicinity of the sudden expansion (section S_1 at $x_1 = 1.0\,m$). In the upstream pipe the flow is fully developped, then the singularity causes abrupt flow separation, which causes recirculation zones. In the downstream pipe, the flow reattaches at Section S_2 and reaches fully developed regime.

This analysis will be performed with the ANSYS WorkBench platform, which integrates the following softwares:

- Geometry creation (DesignModeler)
- Mesh generation (Ansys Meshing)
- Flow setup and numerical solution (Fluent)

[1]The pdf version of this document and all necessary documents for completing the exercise are available online at the course webpage: *UE 203 - Computational Fluid Dynamics M1 MEPP* at the ARCHE website (http://arche.univ-lorraine.fr/enrol/index.php?id=9396").

[2]This lab exercise has been realized with Version 15.0 of ANSYS WorkBench .

Work to be performed

This document will introduce you to the basic functionality of the various WorkBench softwares and will guide you through the CFD analysis of the expansion flow.

Read carefully the document and execute with the WorkBench the tasks presented in the document. You should answer to the various questions of the document in a Word report. You will send me your report at the end of the CFD lectures as part of your evaluation.

2 The WorkBench platform

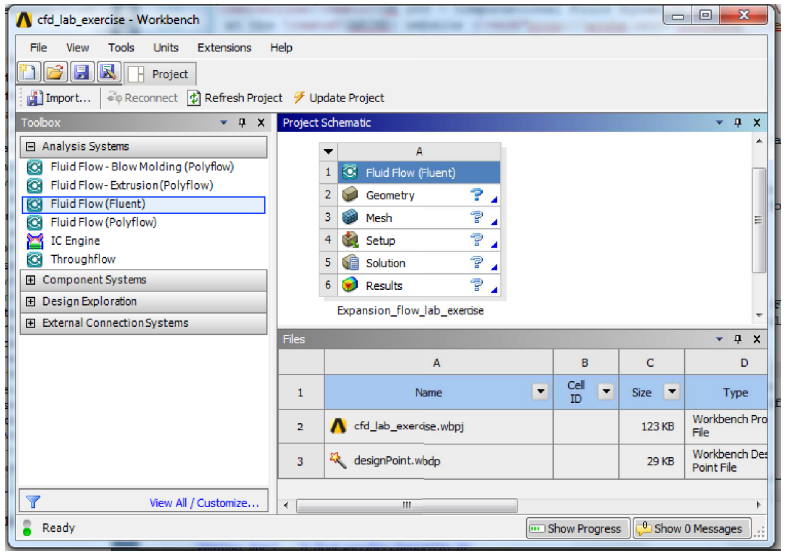


Figure 2: Graphical user interface (GUI) of ANSYS WorkBench .

Launch the ANSYS WorkBench software from your PC and create a Fluid Flow (Fluent) project. The Fluent workflow shown in Fig. 2 shows the consecutive steps of the CFD analysis. WorkBench will automatically manage the various data files generated by your Fluent project.

Save now the WorkBench project (**.wbpj* file) on your disk with a meaningful and correct names for your file and working directory (no special characters, foreign accents, avoid spacings), or the software may malfunction and your data be lost!

The language preferences of the various WorkBench softwares can be modified with the option Regional and Language Options from the menu bar: Tools → Options... You need to restart WorkBench to make the change effective.

3 Computational domain creation with DesignModeler

According to the symmetries of the flow domain of Fig. 1, we assume that the flow is asymmetric so that it is only required to create the computational domain shown in Fig. 3. These assumptions allow us to do 2D computations instead of 3D, that will save computational time and memory space.

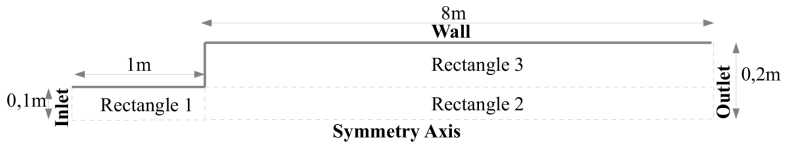


Figure 3: Computational domain and boundary conditions for the cylindrical expansion flow. The 2D computational domain is composed of 3 rectangular surfaces.

For axisymmetric flow computations with ANSYS Fluent , it is important to:

- Create the computational domain in the upper half XY plane (such as $y > 0$) of DesignModeler .
- Ensure that the symmetry axis of the domain is the horizontal axis ($y = 0$).

3.1 Overview of DesignModeler

The graphics interface (GUI) of DesignModeler is shown in Fig. 4. The most relevant components are:

- Graphics Window: sketching area where the various geometrical components of the computational domain are viewed and manipulated with the mouse.
- Menu Bar: where you can read/write the geometry files, create new sketches, *etc...*
- Toolbars: The most important tools are:
  - View toolbars : icons for moving and zooming the geometry with the mouse in the graphics window.
  - Selection toolbars : Icons for mouse selection of the various elements of the geometry (vertex, edge, face and volume) in the graphics window.
  - New sketch : A "sketch" is a 2D drawing of the contours of the computational domain.
- Mode Tabs: there are 2 different modes:
  - Sketching mode: where you draw the contours of the geometry with the help of the various sketching tools shown in Fig. 5.

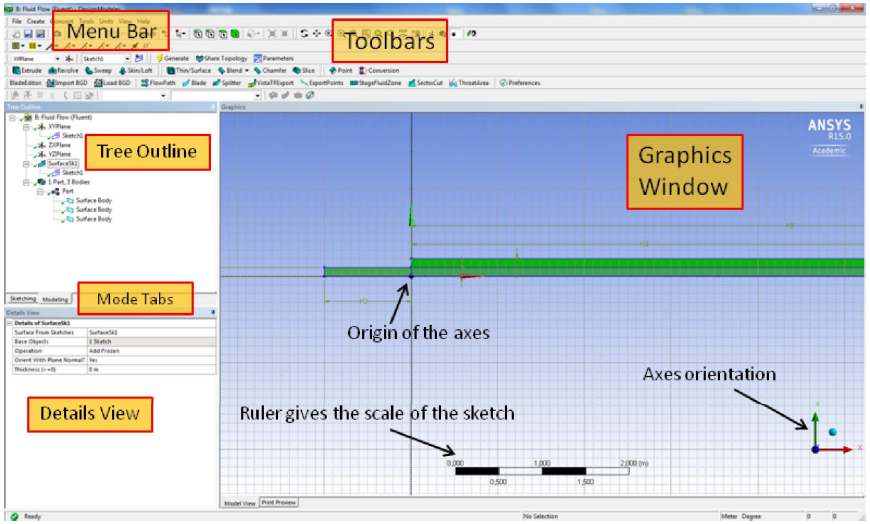


Figure 4: GUI of DesignModeler . Note that 2D geometries should be created in the XY plane. Click on the axes to change orientation of the graphics window.

  - Modeling mode: that shows the Tree outline, where the basic operations for creating the geometry are shown sequentially: the default planes (where the sketches are stored), the geometry operations on the sketches, the list of created parts.
- Details View: where details of the selected geometry (dimensions, constraints, options, *etc...*) are displayed and can be modified. Note that grey colored details cannot be modified.

It is important that you memorize the structure of this GUI, since you will use it extensively to do the lab exercises. Note that the GUI of Ansys Meshing is structured similarly.

3.2 Generation of the computational domain

DesignModeler creates the computational domain with the following consecutive steps:

Step 1) Sketching mode: draw the contours of the domain in one (or several) 2D sketches. It is easier to split the computational domain in several elementary entities (like the 3 rectangles of Fig. 3) and draw them separately.

Step 2) Modeling mode: create "solid" bodies (in 2D, it is a surface body) from these contours.

Step 3) Modeling mode: combine these bodies into a unique computational domain (which is called a part in DesignModeler).

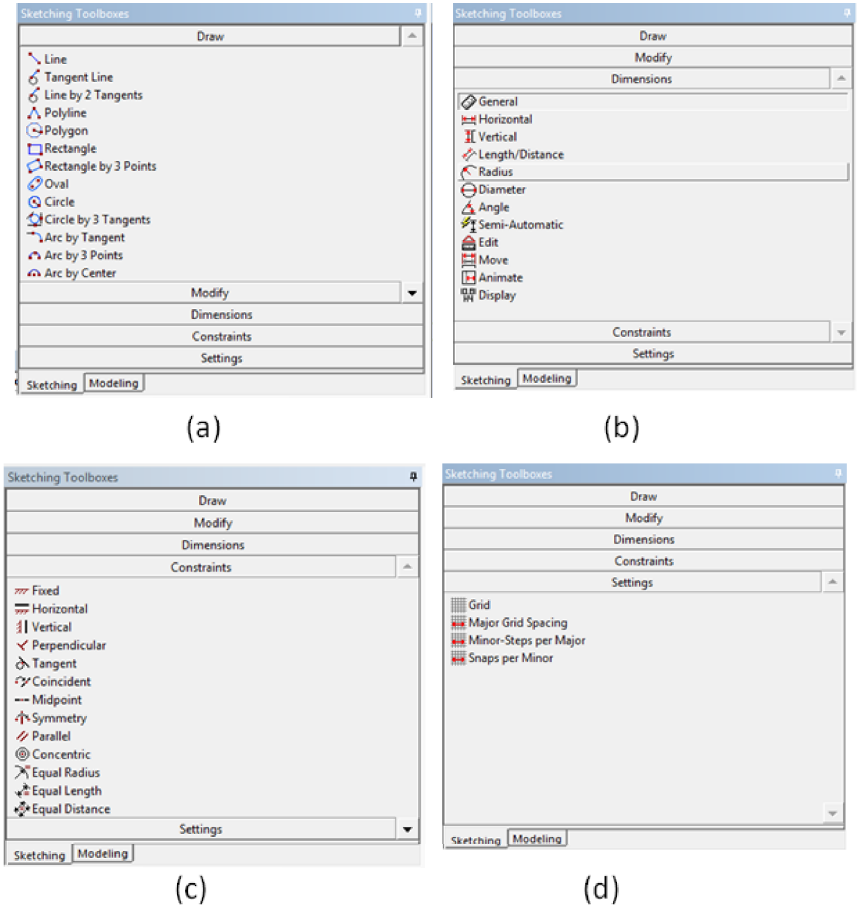


Figure 5: Toolbox available for sketching. (a) Draw: tools for drawing elementary 2D entities (lines, rectangles, circles, ...), (b) Dimensions: tools for setting the dimensions of the entities (by mouse selection in the graphic windows), (c) Constraints: tools for imposing constraints on each entities to assemble the final geometry (alignment with the axes, *etc.*...), (d) Settings: tools for displaying grid lines in the graphic window to make sketching easier.

Apply now this strategy to create the domain of Fig. 4. In one sketch, you will draw separately the 3 rectangles shown in this figure, then assemble them as a unique part with 3 distinct surface bodies.

Note that the Undo/Redo tools are only available in the sketching mode. In the Modeling Mode, you can either Suppress a body[3] or Delete it permanently from the tree outline.

Details View

Details of SurfaceSk1	
Surface From Sketches	SurfaceSk1
Base Objects	1 Sketch
Operation	Add Frozen
Orient With Plane Normal?	Yes
Thickness (>=0)	0 m

Figure 6: Base Objects: the 3 selected rectangles. Add Frozen (in French: Ajouter Corps Bloqué): it means that the three surfaces remain independent in the computational domain. You will be able to mesh each surface independantly in Ansys Meshing . Note that grey colored details cannot be modified.

0) In the Tree Outline of the modeling mode, click on XYPlane and create a new sketch . Go to the Settings toolbox (Fig. 5 (d)) to show the grid and apply adequate parameters for viewing the geometry: Major Grid spacing: 1 m, Minor-steps per Major: 10.

1) In the sketching mode, create Rectangle 1 with the Draw toolbox of Fig. 3 and set its dimensions[4]. Repeat for the other 2 rectangles. Align them as shown in Fig. 4 by using the Constraints toolbox[5].

2) In the Tree Outline of the modeling mode, select the sketch and click on Concept → Surfaces From Sketches from the menu bar. A surface sketch SurfaceSk1 is created in the tree outline, whose details are shown on Fig. 6. To finalize the operation click on Generate .

3) You should have created 3 parts made from the 3 surface bodies you have drawn (look in the Tree Outline). Now, to assemble the 3 bodies in the same part, select them all and right-click on Form New Part. As a result, you have created the computational domain as 1 part made of 3 surfaces bodies, as in the tree outline of Fig. 4.

4) The last step is to define the computational domain as "fluid". To do this, set Fluid/solid to be Fluid in the Details View of the part.

If an element is highlighted with a yellow bolt in WorkBench , it means that the software waits for the user to update/generate this element.

[3] It will be ignored when the geometry is generated.
[4] Define them with Dimensions toolbox and set their values in the Detail View.
[5] Use the Coincident constraint to stitch the edges of each rectangle to the X or Y axes.

Question 1. *Once the computational domain has been created, display it in your* Word *report and exit* DesignModeler .

When you exit the software, the computational domain is automatically saved on disk in the DesignModeler database format (**.agdb*), and ready to be used by the next software in the WorkBench workflow. You can view all files generated during the lab exercise (and their location on your hard-disk) by toggling View → Files in the WorkBench menu bar.

4 Mesh generation with Ansys Meshing

Open now Ansys Meshing from the WorkBench . The GUI, shown in Fig. 7, is similar to DesignModeler with the same mouse functionality. In this first lab exercise, we will only consider the generation of the most simple case of quadrilateral mesh, *i.e.* the Cartesian mesh (see the CFD lectures). Ansys Meshing is able to generate a mesh automatically, with limited user's input. For example, after clicking on the Generate Mesh tool, you should have to create the uniform mesh shown in Fig. 7. The only user input needed is the cells' edge size (Max Face Size in the Details View of the mesh).

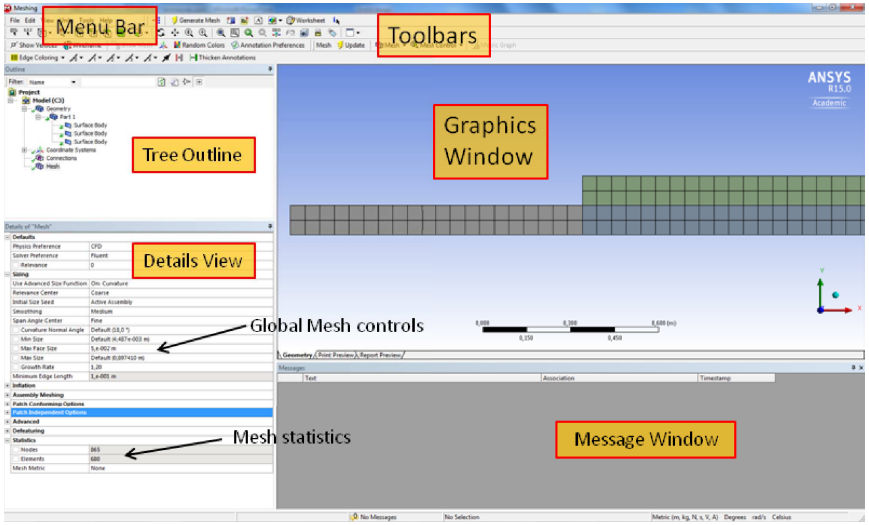


Figure 7: GUI of Ansys Meshing . The quad mesh shown in the graphics window has been generated with Max Face Size=0.05 m and all other parameters set to default values.

Question 2. *Try generate meshes with various cell sizes. Display the resulting meshes in your* Word *report with the corresponding number of cells (*Elements*) and vertices (*Nodes*) given in the* Statistics *detail view of the mesh.*

4.1 Generation of a graded Cartesian mesh

The flaws of the "automatic" mesh of Fig. 7 is that the user does not control the mesh resolution in the vicinity of strong flow gradients, which are located in the boundary layers along solid walls and in the recirculation zone (see Fig. 1). Ansys Meshing allows various strategies for refining locally the mesh in selected areas of the computational domain. In this lab exercise, we will use the "bottom up" strategy to create a better quality mesh: first introduce a 1D meshing of the edges of the domain, then use a quad method to mesh the computational domain (which is called a Face, since it is 2D). The resulting mesh, shown in Fig. 8, is generated by the following steps:

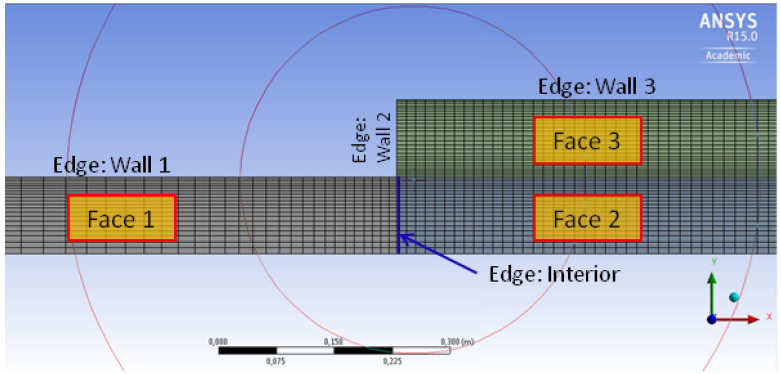


Figure 8: Zoom of the mesh near the singular expansion. Note that the mesh is refined near the walls and the salient corner. This mesh has 14300 cells.

1) Edge meshing (Tool for edge selection): In the Graphics Window, select one of the edges of Face 1 and insert an edge sizing method[6]. The Edge Sizing method appears in the Tree Outline of the mesh. Choose Type: Number of Divisions and enter the details given in Table 1. Repeat for the 3 other edges of Face 1.

Edge	Interior	Wall 1	Wall 2	Wall 3
# of divisions	20	40	25	300
Bias type				
Bias factor	3	5	2,5	4

Table 1: Sizing details for the edges shown in Fig. 8. Number of Divisions corresponds to the number of edge vertices N. Bias Type enables to create a graded mesh with geometric refinement of the form $\Delta x_{i+1} = R\Delta x_i$, where Bias Growth Rate R is the ratio between adjacent cells. Examples of graded meshes are given in Fig. 9. Bias Option: the option Smooth Transition needs the value of R, otherwise Bias Factor corresponds to $\Delta x_{\max}/\Delta x_{\min} = R^{1/(N-1)}$.

[6] Right-click on the moose: Insert → Sizing.

Figure 9: Exemples of graded mesh with geometric refinement. At left: the mesh is refined on the west boundary only. At right: refinement at both west and east boundaries.

2) Face meshing (Tool for face selection:) Complete the meshing of Face 1 by inserting a Mapped Face Meshing with Method: Quadrilaterals. Generate the mesh of Face 1 by right-clicking on Generate Mesh on Selected Bodies.

3) Preview the mesh by clicking on Mesh in the Tree Outline. If the mesh density is not similar to Fig. 8 or the grid lines are not parallel to the axes, you certainly made a mistake in some edge sizing details. In this case, delete the mesh[7], review your edge sizings and generate again the mesh of Face 1.

4) Repeat these steps for Face 2 and 3.

4.2 Definition of the boundary conditions

Now that the meshing of the computational domain is completed, the last step before starting the CFD analysis with the Fluent solver is to label the boundary conditions as in Fig. 3. To perform this, click on Model in the Tree Outline, then select a boundary edge in the Graphics Window and insert a Named Selection[8]. In the Tree Outline, rename it as shown in Fig. 3. The label of the boundary edges will be recognized by Fluent and the corresponding boundary condition will be applied.

For an easy reviewing of the boundary conditions in Fluent by the user (think that an actual 3D mesh may have hundreds of boundary edges), it is worthwhile to group all edges with the same boundary conditions (*e.g.* the 3 edge walls) in a single named selection.

Question 3. *Once the mesh is created and the boundary conditions labeled, display the mesh in your* Word *report, give the mesh statistics and exit* Ansys Meshing .

5 CFD analysis with Fluent

Open Fluent from WorkBench . The Fluent launcher panel will open. Select Dimension: 2D, Options: Double Precision and leave the default settings for the other options. Note that:

- Double precision: if you choose this option, floating point operations will be made on 8 bits, which yields numerical results with a 16-digit precision. Using the default Single Precision (4 bits) will reduce the memory usage and yield faster computations, but will give less accurate results (8-digit accuracy) and the convergence of the iterative solution may be impacted. It is thus recommended to use the Double Precision solver.

[7]Right-click on Mesh in the Tree Outline and select Clear Generated Data. This operation only deletes the mesh, not the meshing methods you have defined.

[8]Right-click on the moose: Insert → Named Selection.

- Serial/Parallel: if your PC has a single processor, choose Serial (the default). If your PC has several processors (or cores), Parallel computations amounts to partitioning the mesh such that the computation on each mesh part is performed by a different processor. The interest of parallel computations is that the computational time of a serial computation is (ideally) divided by the number of processors used.

5.1 Overview of Fluent

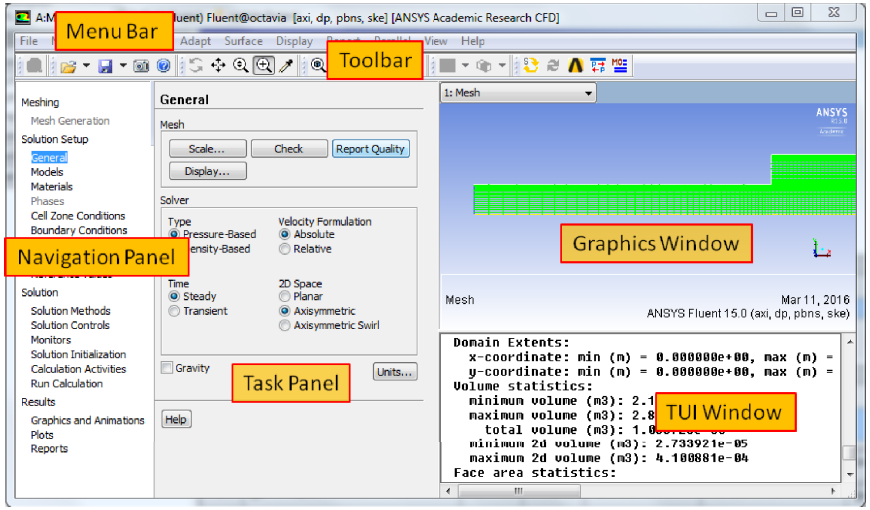


Figure 10: GUI of Fluent .

The graphics interface (GUI) of Fluent is shown in Fig. 10. The most relevant components are:

- Toolbars: As in DesignModeler and Ansys Meshing .
- Graphics Window: Graphical results are displayed here. The default mouse button functionality is[9]:
  - Left button translates.
  - Middle button zooms.
  - Right button selects/probes.
- Navigation Panel: The main stages of the CFD analysis are displayed here as items, organized as a tree from top to bottom:

[9]Note that they are different from the other softwares in WorkBench . They can be changed from Display → Mouse buttons... in the menu bar.

  - Solution Setup: define the CFD problem, such as the fluid models (laminar, turbulent, heat transfer, multiphase, ...), type of materials (water, gas, solid, ...), boundary conditions.
  - Solution: define the numerical inputs (such as discretization schemes, solution algorithms, initial condition, ...) and solve the problem.
  - Results: once the solution is calculated, analyze (postprocess) the results.
- Task Panel: Selecting an item in the Navigation Panel opens the corresponding dialog box in this panel.
- Menu Bar: Additional items can be accessed from the menu bar.
- TUI[10] Window: display text results of the computation.[11]

5.2 Input/output files

There are 2 types of data files generated by Fluent .

- Case File *(*.cas)* : stores the mesh and the setup parameters.
- Data File *(*.dat)* : stores the last computed solution.

You can manage these files from the menu bar File → Import or Export. You can also import a mesh generated from a third-party meshing software. Since you are using WorkBench , the mesh generated previously with Ansys Meshing is imported at the start of Fluent , while Case and Data files are automatically managed when you start or exit Fluent .

5.3 Solution setup

All the Fluent tasks for set-up of the CFD problem are done in the Solution Setup panel. Fluent has predefined default settings for the various stages of the set-up, but it is important to set them correctly for the problem under consideration. Thus, it is important to have knowledge of the physics involved in your CFD problem for selecting the relevant flow simplifications, models and equations to be solved.

Question 4. *We consider the flow of water ($\rho = 998.2\,kg/m^3$, $\mu = 0.001003\,kg/ms$) with inlet velocity $u_{inlet} = 1.5 \times 10^{-3}\,m/s$. Compute the Reynolds number in the upstream pipe (based on its diameter). Infer the flow regime and the relevant flow simplifications.*

Check the mesh and select the CFD solver: General task page

It is important to check the suitability of the imported mesh for Fluent computations. Do it by clicking on Check on the Mesh subpanel. It is also possible to rescale the mesh.

[10] TUI: Text User Interface.

[11] For advanced user: all the tasks defined from graphic panels have a corresponding TUI command, in order to be able to run Fluent in batch mode for automating the solver set up/solution/postprocess stages with scripts and journal files.

As noted in the lectures, there is no universal numerical method that works for all flow regimes: incompressible flow are computed with the Pressure-based Solver, while the Density-Based Solver is used for compressible flows. The other options are related to the problem simplification mentioned in Section 1: Time: Steady, 2D Space: Axisymmetric.

Set up the flow physics: Models task page

The Pressure-based solver is configured with the relevant flow physics in the Models task page: turbulence models, flow with heat transfer (Energy, Radiation), combustion (Species), multiple phases (Multiphase), ... It is of the upmost importance to set up the form of the viscous tensor in the Navier-Stokes equations (Viscous item). Click on the Edit... button to review the various Viscous options:

- Inviscid : viscosity is assumed to be zero, which corresponds to the Euler equations, mainly used for external compressible flows.
- Laminar : corresponds to the classical Navier-Stokes equations with the Cauchy stress tensor. The form of the viscosity (constant, non-Newtonian or temperature dependent) is defined later with the material properties of the fluid.
- Other options are used for turbulent computations, and correspond to choosing the modified viscous tensor resulting from RANS[12] modeling of the Navier-Stokes equations. We can mention the well-know k-epsilon model which is widely used in CFD: using this model will add 2 transport equations (for the turbulent kinetic energy k and dissipation ϵ) to the NS equations. The parameters for these transport equations can be set from these this panel.

By default, Fluent uses the laminar model with all other models (Multiphase, Energy, *etc.*...) disabled.

Set up the material properties: Materials task page

Materials refer to the various fluids or solids involved in your CFD problem. The materials available for your simulation are listed on the Materials task page. By default, Fluent selects the Fluid material as air with constant properties (density and viscosity). Click on Create/Edit... to add water-liquid to the list of available materials (use the Fluent Database, where a wide range of materials are predefined). You can edit the material properties if needed, for example if you wish to use a non-Newtonian fluid with shear-thinning viscosity.

Set up the cell zone conditions

As stated in the lectures, a zone corresponds to a portion of the mesh defined as fluid, solid or porous, where distinct materials can be defined[13]. These zones have been defined previously in DesignModeler : in this first lab exercise, there is only one fluid zone filled with water. Click on

[12] *RANS : Reynolds Averaged Navier-Stokes Equations*, obtained through statistical modeling of turbulent flows.

[13] As an example, the computational domain for a conjugate heat transfer problem would be divided into fluid and solid zones.

Edit... in the Cell Zone Conditions task page to set this zone as water-liquid (by default Fluent fills all fluid zone with air).

For advanced users: In the Fluent pressure-based solver, the computed pressure (the Gauge Pressure p_{gauge}) is defined up to a constant level (the Operating Pressure $p_{\text{operating}}$), such that:

$$p(\boldsymbol{x}) = p_{\text{operating}} + p_{\text{gauge}}(\boldsymbol{x}). \tag{1}$$

The operating pressure can be set at a reference location in the domain by clicking on Operating Conditions... . By default, Fluent sets $p_{\text{operating}}$ as the ambient pressure at the origin of the computational domain, and it is usually not necessary to change it. In addition, if it is required by your CFD problem, you can add gravity to your model with this item.

Note that the value of the operating pressure has no influence on the computed results since only pressure differences are relevant in incompressible flows.

Set up the boundary conditions (BCs)

This task page defines the BCs settings at the boundary edges (in 2D) of the computational domain. The Zone panel displays the name of the boundary edges as you previously labelled them in Ansys Meshing [14]. You can visualize theses edges by clicking on Display Mesh... . Fluent usually recognizes these labels and assigns them with a corresponding BC Type. Make sure the the inlet boundary is defined as velocity-inlet, the symmetry axis as axis, and the outlet as pressure-outlet (see Fig 1). Some BCs needs additional settings which are defined by clicking on Edit... . For this lab exercise, you only have to set the inlet velocity given by Question 1.

5.4 Set up the numerical parameters of the computation

Now that the physical model, solver and boundary conditions have been set, the next step is to define the numerical parameters (algorithm for pressure-velocity coupling, spatial discretization, convergence criteria) of the pressure-based solver: this is performed with the Solution panel.

By default, Fluent chooses "robusts" numerical parameters, which yield converged solutions for a wide range of flows. Henceforth, the default discretization methods are usually low-order and dissipative (for stabilizing the solution) and should be modified to get accurate results.

Set up the discretization methods: Solution methods task page

In this task page (shown in Fig 11) you first set up the iterative algorithm for solving the pressure-velocity coupling (SIMPLE[15] algorithm and its variants, see lecture notes). Use SIMPLE for steady problems such as this lab exercise (PISO is recommended for transient problems). Define next the various finite-volume (FV) discretization schemes for the NS equations [16] (Gradient: cell-face interpolation of velocity gradient used in the discretization of the viscous term and high-order discretization of the convection term, Momentum: discretization of convection term, Pressure: interpolation of cell-face pressure). The various pros and cons of each scheme has been discussed in the lecture notes. As a reminder:

[14] If some of the boundary edges are missing, it may be caused by a bug in the WorkBench . Ask advice to your instructor.

[15] Acronym for *Semi-Implicit Method for Pressure-Linked Equations.*

[16] When applicable, discretization schemes of the additional transport equations need to be defined too. Examples will be given for the turbulent computation of Section 6.

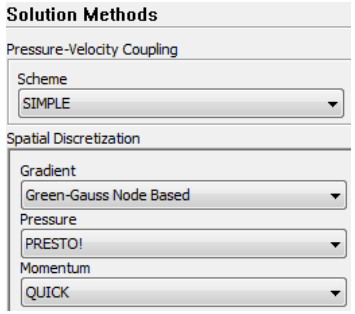


Figure 11: Recommended discretization methods for the expansion flow.

- Use Green-Gauss Node-Based interpolation for meshes with only tri/tet cells.
- Use PRESTO! for pressure interpolation or the solution may have convergence problems.
- For convection, use First Order Upwind only at the initial stages of the computation or if the solution exhibits convergence problems. Otherwise, second-order schemes such as QUICK (more accurate and less diffusive) are preferred.

Set up the relaxation parameters of the iterative algorithm: Solution Controls task page

Background on iterative algorithms for solving a system of algebraic equations: for a (non-) linear system of the form:

$$\mathcal{A}\Phi = RHS, \tag{2}$$

where Φ is the vector containing velocity or pressure mesh values, the matrix $\mathcal{A}$ and vector RHS represent the FV discretization of the NS equation. Starting from the initial condition Φ^0 given by the user, an iterative algorithm seeks the solution as the series $\{\Phi^k,\, k \geq 1\}$ such that:

$$\Phi^k = \Phi^{k-1} + \alpha\, \delta\Phi, \tag{3}$$

where k is the iteration number and $\alpha \in [0, 1]$ is the Under-Relaxation Factor. The value of α, chosen by the user, has impact on the rate of convergence of the algorithm. The closer is α to 1 the faster is the convergence, but non-convergence (or solution blow-up) may occur because the lag $\alpha\, \delta\Phi$ is too large. In the latter case, the user should decrease (*i.e.* , under-relax) the value of α to get convergence, even though it may slow down the algorithm.

The under-relaxation factors are set-up in the Solution Controls task page for each flow equation. The default values given by Fluent are *robusts*, *i.e.* they give a solution for a wide range of problems. There is no need to modify them unless your computation undergoes convergence problems.

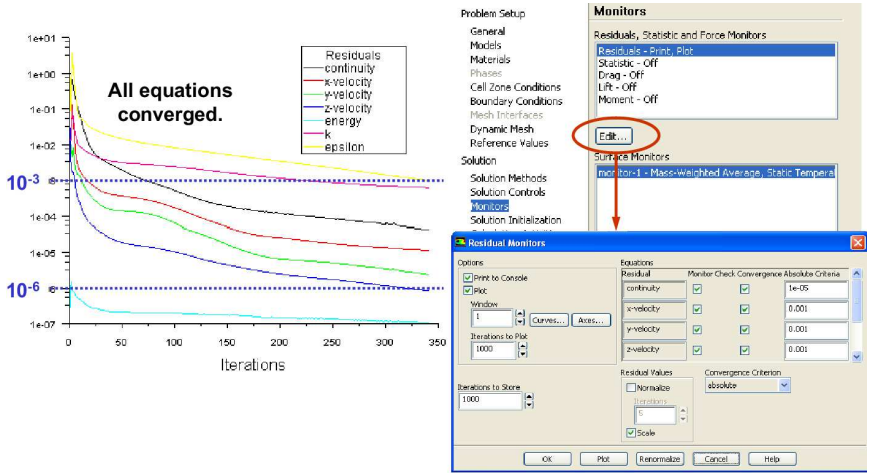


Figure 12: Example of residual monitoring plot for a turbulent flow computation.

Monitoring the convergence of the iterative algorithm: Solution controls task page

Fluent considers that the iterative algorithm has converged when the residual R^k of Eq. 2, defined by $R^k = RHS - \mathcal{A}\Phi^k$, falls below the convergence criterion:

$$\|R^k\| < \epsilon, \tag{4}$$

where the value of the tolerance threshold ϵ is given by the user.

The set up of the convergence monitors for each scalar equation of the NS system is performed in the Monitors task page. By default, Fluent defines the convergence criteria to be $\epsilon = 10^{-3}$ for all flow equations[17]. This value is usually not low enough to give accurate results. To change this criteria, click on Edit... in the Residuals panel (see Fig. 12). During the computation, the evolution of the residuals will be shown in the graphics window as in Fig. 12 if you choose the Plot option.

5.5 Perform the computation

Computations are performed in the Solution panel. You need first to define the initial condition Φ^0 of the iterative algorithm[18]. This is performed in the Solution Initialization task page. Fluent proposes several Initialization Methods for defining suitable initial conditions, notably from the values defined at the boundary conditions. For this lab exercise, you will start the algorithm from rest ($\boldsymbol{v} = 0$, $p = 0$). To do this, choose Standard Initialization in the task page, Compute From: all-zones, and press Initialize.

[17] Except the Energy equation for heat transfer problems where it is set as $\epsilon = 10^{-6}$.

[18] For unsteady problems, the initial condition corresponds to the solution at initial time $t = 0$.

To start the iterative algorithm, go to the Run Calculation task panel, enter Number of Iterations: 10 and press Calculate. As a result, Fluent will write (in the TUI) and plot (in the graphics window) the residuals at each iterations[19]. Since after only 10 iterations the convergence criteria (4) is not met yet, set a larger number of iterations (say 1000) and click on Calculate to continue the iterative algorithm: Fluent will automatically stops the algorithm when convergence is reached.

Note that Fluent stores in memory the solution at the last iteration only. If you have pressed Calculate again, you will continue the algorithm from the latest saved iteration, *i.e.* the tenth. If you want to restart the algorithm from rest, you should do the Solution Initialization step first before clicking on Calculate.

You are now able to perform by yourself a full computation of the flow:

Question 5. *Perform a computation of the expansion flow with the convergence threshold $\epsilon = 10^{-5}$ for all flow equations. Display the residual monitoring plot in your* Word *report*[a]. *Save the converged solution to your PC disk (see Section 5.2 for details): give the name* water_uinlet=0,0015 *to the* *.cas *and* *.dat *files.*

[a] For exporting a Fluent plot, right-click at the top of the graphics window and select Copy to Clipboard.

For advanced users: to perform periodically a back-up copy of the solution during the iterative algorithm, go to the Calculation Activities panel and enter the frequency back-up in the Autosave Every (Iterations) group box.

5.6 Postprocessing: Results panel

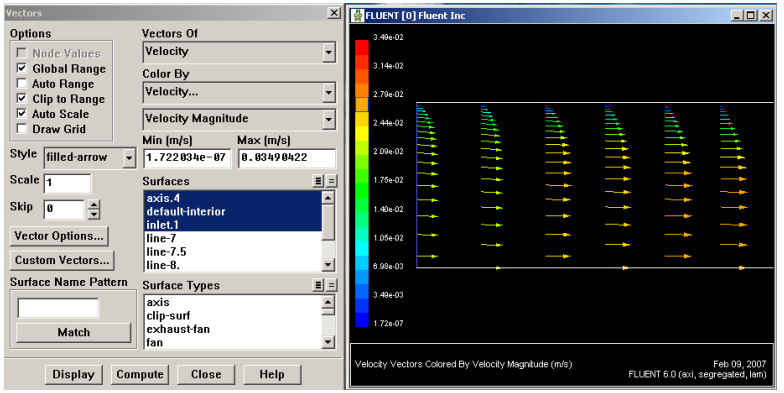


Figure 13: At left: Vectors dialog box. At right: 2D vizualisation of the velocity vectors upstream of the contraction.

[19] The frequency of the residual plots and writes can be changed with the Reporting Interval option.

Once the converged solution is obtained, Fluent proposes numerous postprocessing tools to analyze the solution in order to extract relevant flow information. This is performed under the Results panel. There are 3 main types of analysis:

- 2D (or 3D) solution visualization (Graphics and Animation task panel): for plotting the velocity vectors, contour levels of a flow variable (velocity, pressure) inside the computational domain.
- 1D visualization (Plots task panel): for profile plotting of the solution on selected sections (or lines) of the computional domain.
- Numerical reporting of mean flow quantities (Reports task panel): such as mass flow through selected sections, forces at boundaries, ...

Velocity vector visualization: Graphics and Animation task panel

Choose Vectors in the Graphics group box and click on Set Up... . The Vectors dialog box of Fig. 13 (left) opens. From the Surfaces drop-down list choose interior to display the velocity vectors in the whole computational domain[20], and press Display. You can use the mouse to zoom in/out in the graphics window.

Question 6. *Zoom in the computational domain to observe the recirculation zone of the flow (see Fig. 1). Use the Scale options of the Vectors dialog box if the vectors are too small. Display this plot in your Word report.*

Isocontour visualization: Graphics and Animation task panel

Choose Contours in the Graphics group box and click on Set Up... to open the Contours dialog box. Select a flow variable in the Contours of drop-down list and press Display.

Most flow variables are predefined in Fluent with intuitive names. For example, for asymmetric flow:

$$u = \text{Axial Velocity}, \qquad v = \text{Radial Velocity}, \qquad ||\boldsymbol{v}|| = \text{Velocity Magnitude}.$$

For pressure, note that Static Pressure and Absolute Pressure correspond respectively to p_{gauge} and p in Eq. (1), while the Total Pressure is :

$$p_{\text{total}} = p_{\text{gauge}} + \tfrac{1}{2}\rho\,||\boldsymbol{v}||^2. \tag{5}$$

By default, Fluent plots 20 iso-values between the min. and max. values of the visualized variable. In the Contours dialog box, you can set the number of levels with the Levels option, and modify the min. and max. levels by deselecting the Auto Range option and choosing manually the Min and Max values.

Question 7. *Display the isobaric lines of the flow near the contraction. Export the graphics to your Word report. At what distance downstream of the singularity we get the isobaric lines of the Hagen-Poiseuille flow in a straight pipe ?*

[20] Alternatively, you can visualize the flow on selected surfaces and boundaries from the Surfaces drop-down list.

Velocity profiles on sections of the computational domain

You will learn now how to plot velocity vectors on selected surfaces (lines in 2D) of the computational domain, for example on the $x = 7\,\text{m}$ section[21] near the outlet boundary, in order to verify that the flow is full developed there. To do this, select Surface → Line/Rake... from the menu bar to create the line $x_0 = 7.0$, $y_0 = 0$, $x_1 = 7.0$, $y_1 = 0.2$. Give the line a meaningful label, such as line-7.0 in New Surface Name. The surface line-7.0 will now be available in the Surfaces drop-down list of any Graphics and Vizualization dialog box.

Question 8. *Plot the velocity profiles on Sections $x = 6.0, 6.5, 7., 7.5$ and* pressure_outlet.* *(Give them a meaningful name!). Check that the Poiseuille velocity profile is obtained on these sections. Export the graphics to your Word report.*

Plot 1D profiles

The plotting of 1D profiles of predefined flow variables (axial or radial velocity, static or total pressure, ...) is performed on the Plots task panel. Choose XY Plot in the Plots group box and click on Set Up... to open the Solution XY Plot dialog box. For plotting along the radial direction y, choose the direction vector X = 0, Y = 1, Z = 0 in the Plot Direction group box.

Question 9. *Plot the velocity and (static) pressure along the sections of Question 8. Export the graphics to your Word report. Do these profiles correspond to a fully developed Poiseuille flow ?*

Define custom field functions

It is possible to define new flow variables (named "custom field functions") from Fluent predefined variables, by using the "pocket calculator" provided by Define → Custom Field Functions... from the menu bar. This new variable can be visualized as previously.

Question 10. *Define the new function* tau_xy *that calculates the shear stress in cylindrical coordinates :*

$$\tau_{xy} = \mu\Big(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\Big).$$

Plot the shear stress profile on the vertical sections of Question 8 and along the upper wall. Export the graphics to your Word report.

Calculate mean values

For example, you can calculate the mean flow velocity in Section $x = 8\,\text{m}$ of the pipe. In the Reports task panel, open the Surface Integrals dialog box and choose Area Weighted Average in the Report Type drop-down list.

Question 11. *Verify that the mean flow velocity is constant on the sections of Question 8. Report these values in your Word report.*

[21] Remember that the origin $x = 0$ correspond to the section of the abrupt expansion.

5.7 Application : calculation of standard pressure drop

In a duct of diameter D, the pressure (or head) loss due to viscous friction between sections x_1 and x_2 reads :

$$H_1 - H_2 = \lambda \frac{\overline{V}^2}{2g} \frac{x_2 - x_1}{D}, \tag{6}$$

where λ is the (dimensionless) friction factor, $\overline{V}$ is the mean flow velocity, g is the gravity, $H_i = \overline{p}_{\text{tot}}(x_i)/\rho g$ is the mean head at Section x_i. This formula is valid when the streamlines are parallel to each others, *i.e.* when the flow is fully developed in the pipe.

Question 12. *According to the previous questions, the flow is fully developed between the zones* $6.5 \lesssim x \leq 8$ *of the pipe. Use* Fluent *to calculate the total pressure in various sections of the fully developed region, and give the corresponding value of* λ*. Compare your results with the empiric relations obtained in a straight pipe:*

- *Poiseuille formula for laminar flows,* $Re_D \lesssim 2000$ *:*

$$\lambda = \frac{64}{Re_D},$$

- *Blasius formula for turbulent flows,* $4 \times 10^3 \lesssim Re_D \lesssim 10^5$ *:*

$$\lambda = 0.3164 Re_D^{-1/4}.$$

6 Numerical computations in the turbulent regime

6.1 Performing the computation

We consider now flow of water with inlet velocity $u_{\text{inlet}} = 1.5 \times 10^{-1}\,\text{m/s}$.

Question 13. *Calculate the Reynolds number. Deduce the flow regime and the flow equations to solve.*

If a turbulence model is needed, use k-epsilon Standard. In addition to the averaged Navier-Stokes equations, transport equations for k et ϵ are now solved: you needed to define their solution settings, in particular:

- For inlet BCs, choose Intensity and Hydraulic Diameter in the drop-down list Turbulence Specification Method, and set a turbulent intensity of 5% and an hydraulic diameter of 0.2 m.
- Use 2nd-order discretization for the convective terms of these equations.
- Define the tolerance monitoring parameters for these equations.

Question 14. *Starting from rest, compute the solution with residual threshold* $\epsilon = \times 10^{-6}$ *for all equations. Export the residual history to your* Word *report.*

When performing a high-Reynolds number with impulsive start from the rest, the iterative solution may undergo convergence problems: the residuals do not decrease monotonously, stall or increase until solution blow-up. If this happen, you need to modify the default solver settings for obtaining convergence. The "optimal" parameters for convergence are usually problems dependent, and fine-tuning of these parameters requires expertise and trial-and-errors. General guidelines for convergence troubleshooting are given in the lecture notes. Here is a quick summary:

- Initialize the solution with a previously calculated solution at lower Reynolds number, then gradually increases the inlet velocity.
- Use first-order methods in the initial stages of the iterative algorithm, then use more accurate methods.
- Decreases the values of the under-relaxation factors of the equation(s) that cause(s) difficulties (usually, it is the continuity equation).
- Improve the mesh quality and use finer cells in zones of high flow gradients.

However, for this lab exercise you shouldn't experience convergence problems: After an initial stage where the residuals seem to be stalled, the solution should converge after a few thousand iterations. Note however that this behaviour may depends on the discretization parameters you chose or the version of Fluent you are using.

6.2 Analyze the results

General flow features

Question 15. *Verify that the flow is fully developed at the outlet. Display in the 2D domain the velocity profiles near inlet (several sections in the upstream pipe) and outlet (sections of Question 8). Compare with the laminar velocity profiles obtained previously. Report these results in your* Word *report.*

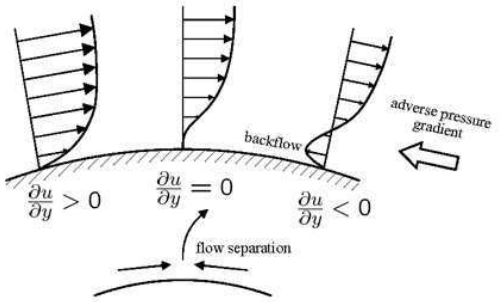


Figure 14: Velocity profiles at a wall boundary layer near a separation point.

Calculation of singular head losses

At the vicinity of a geometrical singularity such as the sudden expansion at $x_1 = 0\,m$ (*cf.* Fig. 1), the flow is no more unidirectional and the head loss equation (6) is not valid anymore. Experiments show that singular head losses are proportional to the mean kinetic energy:

$$\Delta H = \frac{\xi}{2g}\,\overline{V}^2,$$

where ξ is a dimensionless coefficient that depends on the type of singular geometry. For singular expansion flows, the value of the head-loss coefficient ξ can be estimated analytically at high flow rates with the following assumptions:

- Losses due to wall friction are negligible,
- The streamlines are parallel through Section S_1 ($x_1 = 1.0\,m$) and Section S_2 (*cf.* Fig. 1), where the flow returns to fully developed regime. This implies that pressure distribution is uniform at these sections.

With these hypotheses, conservation of mass and momentum between S_1 and S_2 yields:

$$H_1 - H_2 = \frac{\xi}{2g}\,\overline{V}^2(x_1), \qquad \text{with} \quad \xi = \left(1 - \frac{S_1}{S_2}\right)^2. \tag{7}$$

This formula can be found in any fluid dynamics textbook.

Question 16. *From the head losses given by your numerical solution, calculate the head-loss coefficient ξ and compare with the analytical value. The difficulty is to estimate the abscissa x_2 where the streamlines do reattach. To estimate x_2, you may use the property that the shear stress is zero at a separation point (where flow separates or reattaches, see Fig. 14): the value of x_2 is then obtain by plotting the computed shear stress along the upper wall (cf. Question 10).*