Notes

1- Approaches (advantages & disadvantages) اساليب التعامل مع المسائل

- Experimental خاص مضبوط ولكن يحتاج اجهزة وكلفة ودقة واmان ووقت طويل.
- Theoretical عام دقيق ولكن للمشاكل او المسائل البسيطة.
- Numerical (Program or Package) واسع التطبيق وكلفة ووقت وخطورة اقل ولكن معقد وتقريبي.

2- لماذا نستخدم هذه البرامج (Packages)؟

ج: تعلّم برامج رسم وتصميم ومحاكاة متطورة وقابلة للتعديل او التغيير في اي وقت - المعرفة قبل التنفيذ او استحالة التنفيذ – توقع ما حصل او سيحصل – ربط اكثر من عِلم – تعدد الحالات – فهم ما يحصل فعلاً داخل المنظومة – تحوي على كل العلوم التي قمت بدراستها واكثر بكثير.

3- المادة ANSYS 15 / Computer Aided Engineering

- المحاضرات (تفاعلية) داخل المختبر اقل من 30 اسبوع وستكون هناك واجبات (Homework & Classwork)، وتُسمّى نشاط مختبري وتجمع وتقسّم على عدد الواجبات وتسلّم في كل اسبوع، ولا يُقبل التأخير الاّ بعذر رسمي، اما بدون عذر فيُصلّح الواجب البديل من 5 بدل الـ 10، وفي حال رغبة الطالب زيادة الدرجة فيُكلّف بواجب اضافي يُحدد من قبل الاستاذ المشرف.
- توزيع الدرجات:

دروس التقييم المستمر ذات المختبر							
الموضوع	الفصل الأول		الفصل الثاني		امتحان مختبر	السعي	الدرجة النهائية
	امتحان	نشاط مختبري	امتحان	نشاط مختبري			
CAE	10%	10%	10%	10%	10%	50%	50%

- صيغة المحاضرة: Word, Power Point, PDF, Movies, Pictures, Lectures, Books, Examples, Solved Cases, Questions and Solutions, …etc.
- كل هذا موجود على حاسبات المختبر في الـ (C) مع اسئلة السنوات السابقة والواجبات الصفّية والبيتية وكذلك على موقع الخزن IDriveSync اما التبليغات والمستجدات فتكون على الـ Facebook.

4- ما مطلوب من الطالب:

- حاسبة شخصية جيدة وسريعة.
- فلاش رام لنقل اي شيء من حاسبة المختبر.
- تنصيب البرنامج على الحاسبة الشخصية (بدون ربط انترنيت مباشر).
- التواصل عبر الفيسبوك:

أ- ابحث عن الحساب Cae Cad Group من خلال ميزة البحث في فيسبوك وكتابة الايميل التالي: caecadgroup@gmail.com

ب- اضف الحساب كصديق على الفيسبوك مع ارسال رسالة تذكر فيها المرحلة والاختصاص والشعبة واسمك الثلاثي كي يتم التعرف عليك من قائمة الاسماء.

ج- سوف يقوم احد تدريسيي المادة بقبول الاضافة وثم ضمك الى المجموعة.

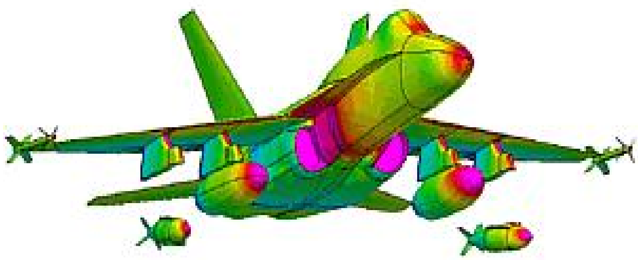
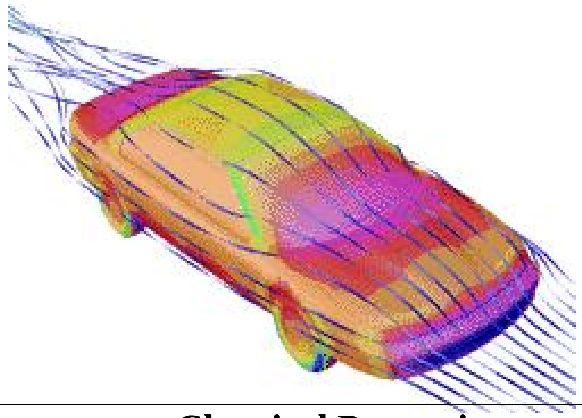
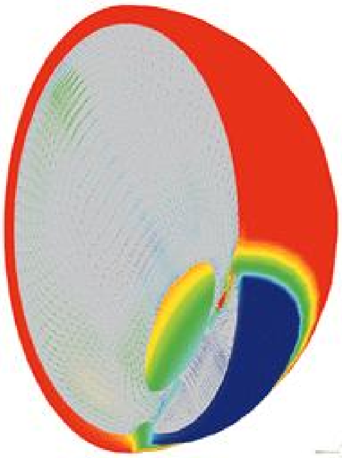
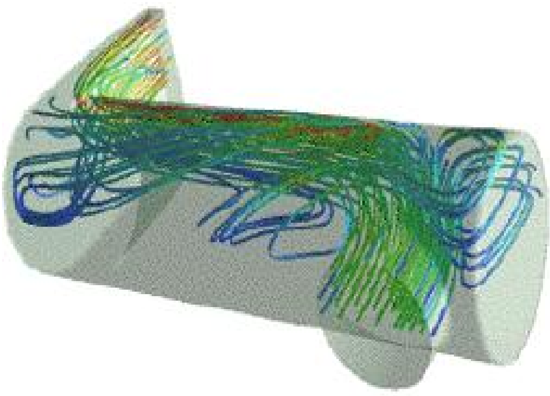
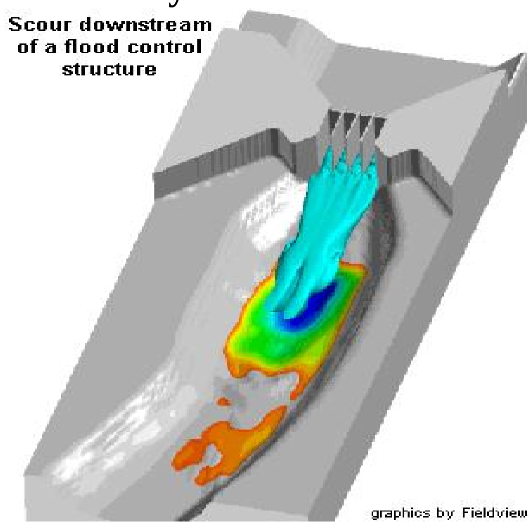
- خزن الواجبات الصفّية والبيتية لكل طالب على حاسبته في المختبر التي تحمل رقمه في قائمة الحضور بإسم project على الـ (D) داخل حافظة تحمل تسلسل واسم الطالب الثلاثي وتكون داخل حافظة بالمرحلة والتخصص والشعبة كلها باللغة الانكليزية ، ويكون هذا هو السياق في الامتحانات ايضاً. كما في المثال التالي:

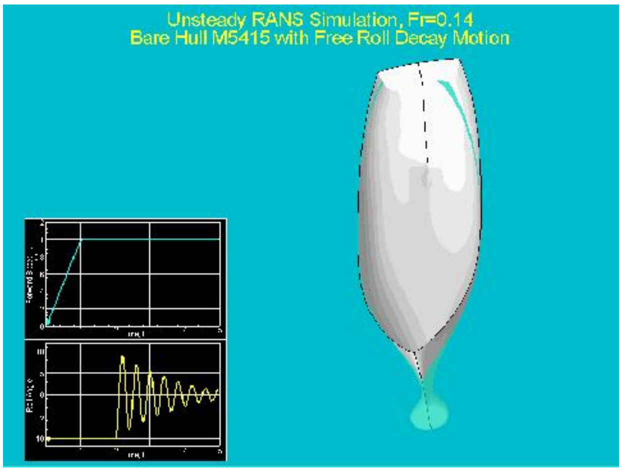
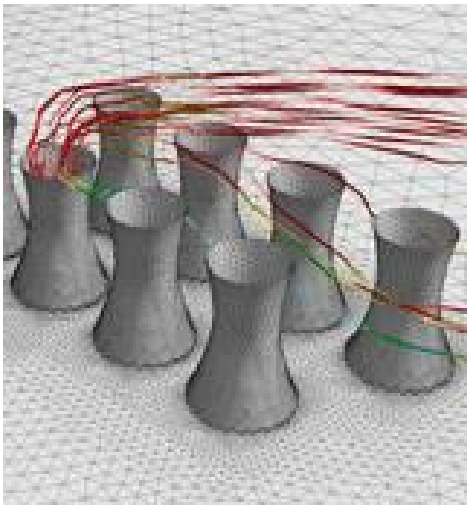
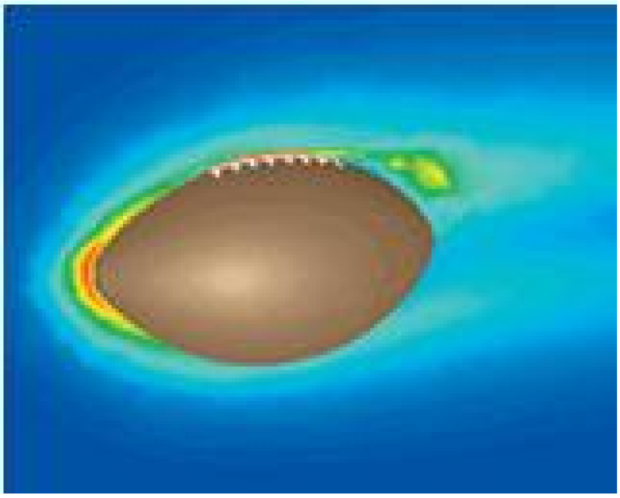
D:\ 4th Year \ GE-A or -B (or AC-A or -B or AU or AE) \ 1Ahmed Mustafa Mohammed \ project1 (HW or CW).

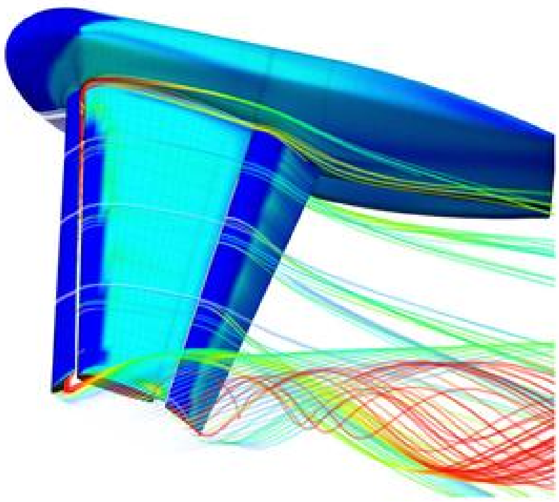
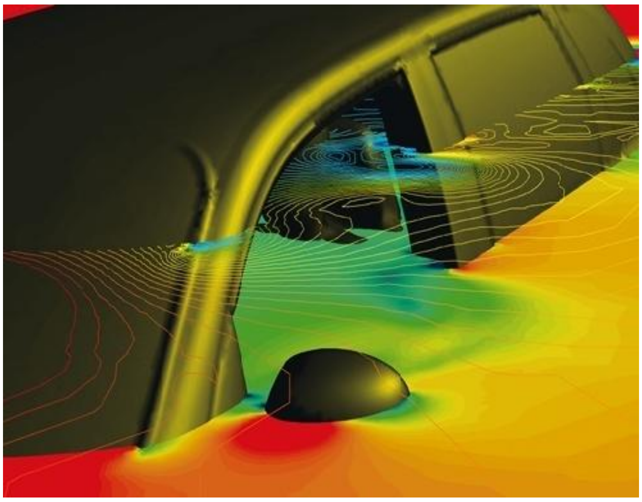
:CFD -5

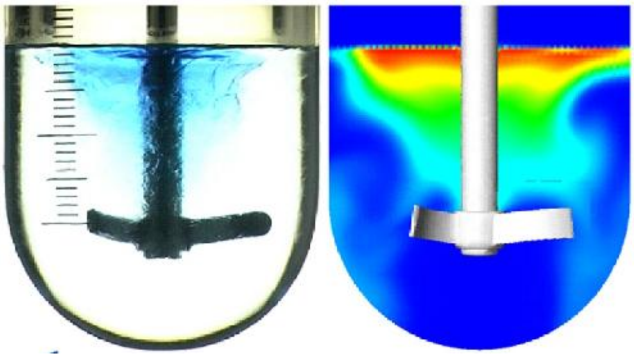
Computational fluid dynamics (CFD) is a tool used to simulate fluid flow and heat transfer problems, using numerical solutions to the equations describing such transport phenomena, for example the Navier Stokes equations which describes the flow of fluids inside a defined flow geometry.

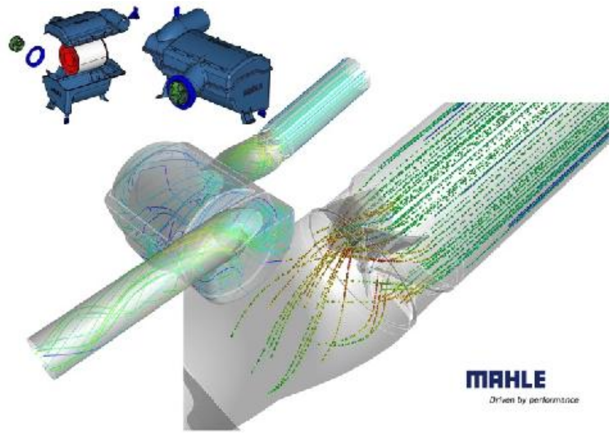
- **Where is CFD used?**

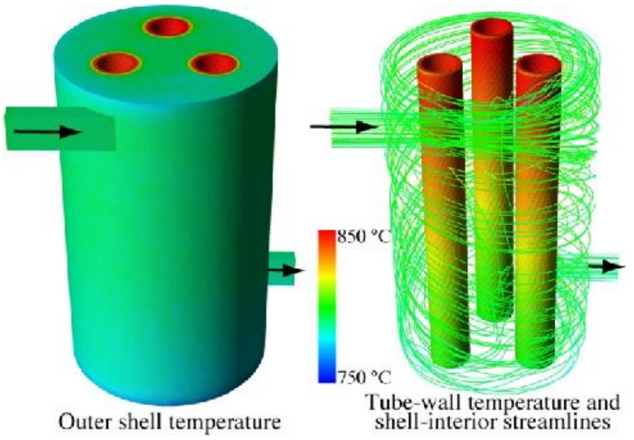
• **Aerospace** 	• **Automotive** 
• **Biomedical** 	• **Chemical Processing** 
• **HVAC (heating, ventilation, and air conditioning)** 	• **Hydraulics** 

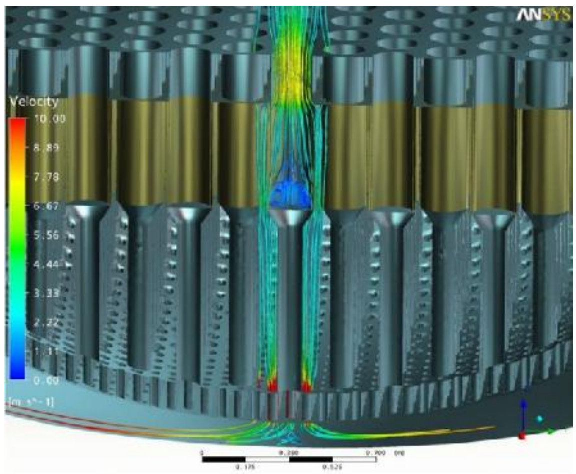
• **Marine** 	• **Oil & Gas** 
• **Power Generation** 	• **Sports** 

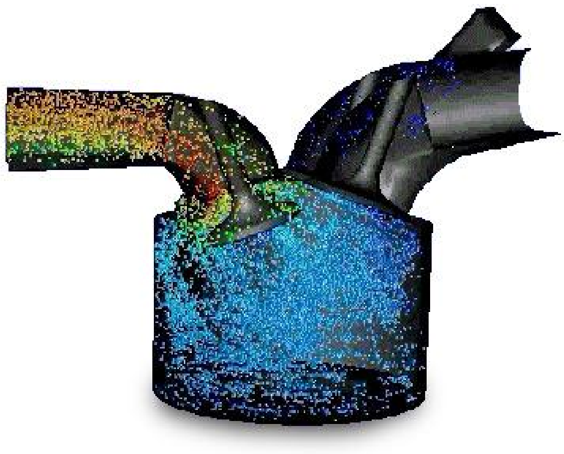
	

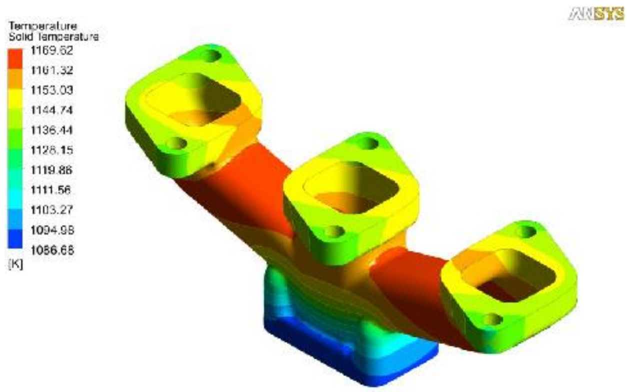


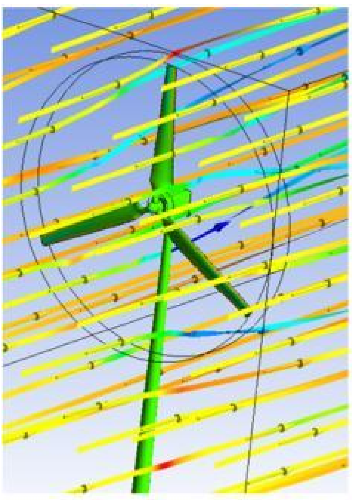

MAHLE
Driven by performance

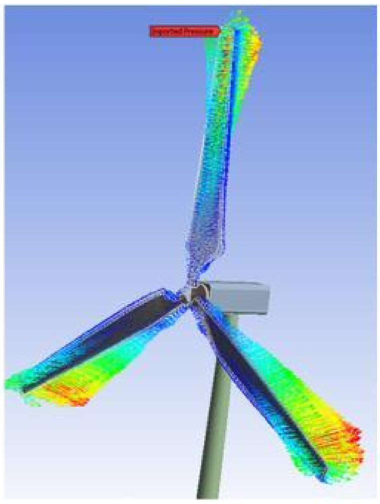

850 °C
750 °C
Outer shell temperature
Tube-wall temperature and shell-interior streamlines

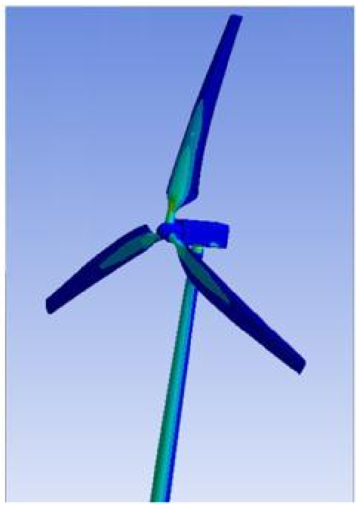

ANSYS
Velocity
10.00
8.89
7.78
6.67
5.56
4.44
3.33

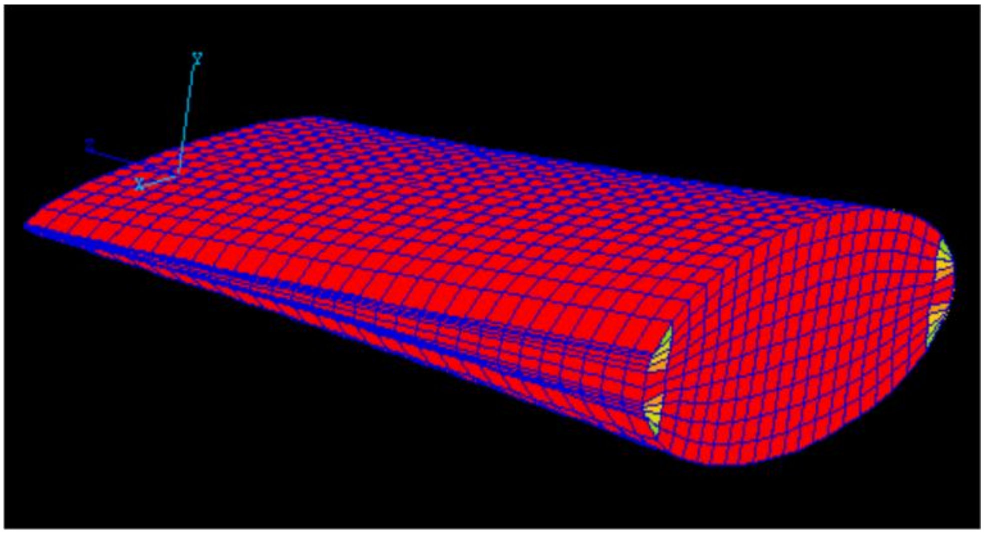



ANSYS
Temperature
Solid Temperature
1169.62
1161.32
1153.03
1144.74
1136.44
1128.15
1119.86
1111.56
1103.27
1094.98
1086.68
[K]









<u>استعراض للحافظات</u>

Summary of the main types of fluid flow problems that general purpose CFD codes can solve:

<u>Types of flow</u>

- **Steady or transient**
- **Viscous or inviscid**
- **Laminar or turbulent lent (using a variety of turbulent models such as the k-ε model)**
- **Compressible or incompressible**
- **Subsonic or supersonic speeds, or ultrasonic**
- **Two phase (continuous phases or particles)**
- **Chemical reacting**
- **Combustion**
- **Swirling**
- **Non-Newtonian**

<u>Modes of heat transfer</u>

- **Convection**
- **Conduction**
- **Radiation**

<u>Types of material</u>

- **Fluid (liquid or gas)**
- **Solid (homogenous or porous)**

<u>Co-ordinate systems</u>

- **Cartesian**
- **Cylindrical polar**

- **Curvilinear**
- **Body fitted**
- **Moving/rotating**

6- التنصيب:

- **خطوات تنصيب برنامج الـ ANSYS يعتمد على الاصدار ، وفي كل اصدار توجد تعليمات خاصة موضّحة بالتفصيل ، وهي عبارة عن صور في الحافظة ANSYS setup داخل حافظة المحاضرات. او اتباع ما يلي:**
- **قبل تنصيب ANSYS يجب ان يكون الانترنيت مغلقاً ، هذا البرنامج مهكّر بسبب ثمنه الباهض**

قم بتنصيب السواقة الوهمية من مجلد Virtual CloneDrive 5.4.2.5 ستظهر سواقة وهمية (BD) قم بالدخول الى الـ DVD وفتح الملف ادناه:

(open with Mount Files with Vertual CloneDrive)

ANSYS.V13 X32 or 64 / DISK1/ m-a1332a

ثم قم بالتنصيب (setup) او بالفتح (open) والتنصيب ، والمهم نقرة يمين و run as administrator .

- **تنصيب البرنامج :**

1- قم بتنصيب (البرامج الضرورية لعمل البرنامج)

Install required prerequisites

وذلك بفتح الـ DVD والبحث عن (Install PreReqs.exe)

2- قم بتنصيب البرنامج

Install ANSYS, Inc. Products

أثناء التنصيب اضغط على Next دائماً وأشّر على skip this step

أكمل التنصيب و اضغط Next سيتأخر التنصيب بسبب الحجم الكبير للبرنامج والبرامج الفرعية مثل الـ Fluent ، انتظر حتى تكتمل الخطوات بعدها ستظهر نافذة تطلب منك القرص الثاني Enter the mount directory: #2 ثم Browse ثم قم بالذهاب الى القرص الثاني (DISK2) واذا لم يفتح فلا تغلق النافذة بل قم بفتح القرص الثاني بشكل اعتيادي من السواقة الوهمية ثم افتح الملف (m-a1332b) وكالمعتاد:

(open with / Mount Files with Vertual CloneDrive) ثم انسخ كل محتويات الحافظة على سطح المكتب مثلاً بعد عمل حافظة جديدة لتكن 123 مثلاً وانتظر حتى ينتهي النسخ ثم ارجع للتنصيب واكمل ولكن هذه المرة من سطح المكتب واختار الحافظة 123 ، وانتظر حتى تنتهي الخطوات الجديدة والخطوات الفرعية ثم Next ثم سيكتمل التنصيب ثم ستظهر نافذتين احداهما Specify the license Sever Machine ابقي على 2325 و1055 والـ 1-server اما الـ Hostname اذهب وافتح:

DISK1/ license / win32 / Shared Files / Licensing / win32 / WinHostId.exe

وهذا WinHostId.exe تنسخه الى الدسك توب وعند تشغيله يكون داخله الـ Hostname & flexId

ثم Exit ثم Next اكثر من مرة ثم اغلق الـ Internet Explorer ثم Finish

وبهذا قد اكتمل تنصيب البرنامج بدون تفعيله اي فقط الـ HELP وفتح بعض نوافذ البرامج.

3- بعدها اكمل التنصيب بترك الخطوة الثانية الوسطية وهي:

Install MPI for FLUENT parallel and Distributed Mechanical APDL (ANSYS)

الاّ اذا كنت تستخدم هذه الامكانية.

4- بعدها وقبل اختيار الخطوة الثالثة وهي تفعيل الـ license يجب عمل الـ license وذلك بالذهاب الى :

DISK1 / MAGNiTUDE / AP13_calc

وهذا AP13_calc تنسخه الى الدسك توب وعند تشغيله اضغط n او y ، اذا ضغطت n فيطلب الـ

Enter target host name افتح WinHostId.exe ثم ادخله بالكتابة ثم Enter ثم Enter target host id كذلك الـ FLEXID ثم Enter ثم Enter

اذا ضغطت y فإنتظر حتى يكتمل انشاء ملف الـ license

فيظهر الـ license على سطح المكتب.

5- اذهب الى الـ دي في دي الخطوة الثالثة واذا كانت غير مفعّلة اذهب الى:

Programs/ANSYS, Inc. License Manager/server ANSLIC_ ADMIN Utility/Run the License Wizard

ثم Continue ثم اختار الـ license من الدسك توب.

وبذلك اصبح البرنامج مفعّل بكل تفرعاته . هذه الخطوة تحتاجها كلما وجدت التالي بعد Run the License Wizard

:ANSYS, Inc. License Manager status

Licensing Interconnect: not running

Licensing Interconnect Monitor: not running

.FLEXlm: not running

لتصبح:

:ANSYS, Inc. License Manager status

Licensing Interconnect: running

Licensing Interconnect Monitor: running

.FLEXlm: running

ملاحظة: ممكن ان يكون التفعيل كالتالي ايضاً:

Programs/ANSYS 13.0/ ANSYS Client Licensing / Client ANSLIC_ADMIN Utility

ستظهر لك نافذة اختر منها

Display the license server machine hostid

ثم اضغط ok

ستظهر عندها معطيات جهازك

بالعودة الى الملف المرفق AP13_calc سيطلب منك ادخال اسم السيرفر

Enter target host name

اكتب اسم السيرفر من نافذة المعطيات ثم اضغط enter

اكتب رقم السيرفر (أيضا من نافذة المعطيات)

Enter target host id

ثم اضغط enter

اضغط أي زر لاغلاق الملف المرفق

سيظهر ملف جديد بجانب الملف المرفق اسمه license.txt

اذهب الى السواقة الليزرية ثم قم بتنصيب

Install ANSYS, Inc. License manager

أكمل الخطوات اللازمة

ستظهر لك نافذة اذهب منها الى سطح المكتب وثم باختيار license.txt

ثم اضغط continue مرتين ثم exit

ستظهر لك نافذة انتهاء التنصيب اضغط finish

ملاحظة: يجب ان يكون هناك Net واحد مربوط في الحاسبة لأن البرنامج سيتعرف عليه.

7- المنهاج:

1- Introduction to CAE

2- Instauration of software, advantages & disadvantages

3- Simulation steps in CAE software& application of CAE in mech. eng.

4- Performing thermal analysis using steady state thermal analysis system.

5- Performing thermal analysis using steady state thermal analysis system. Ex.1 + H.W.

6- Performing thermal analysis using steady state thermal analysis system. . Ex.2 + H.W.

7- Transient state thermal analysis system.

8- Introduction to design modeler: Basic geometry entities

9- Creation of solid model. Ex.1 + H.W.

10- Creation of solid model. Ex.2 + H.W.

11-Configuring CAE & link with external CAD software

12- Importing geometry form external CAD software Ex.1 + H.W.

13- Importing geometry form external CAD software Ex.2 + H.W.

14- Simulation of static structural analysis Ex.1 + H.W.

15- Simulation of static structural analysis Ex.2 + H.W.

16- Simulation of Linear buckling analysis Ex.1 + H.W.

17- Simulation of Linear buckling analysis Ex.2 + H.W.

18- Simulation of modal analysis (free vibration) to extract natural frequency & modes shapes Ex.1 + H.W.

19- Simulation of modal analysis (free vibration) to extract natural frequency & modes shapes Ex.2 + H.W.

20- Simulation of Harmonic analysis (forced vibration) to extract natural frequency & modes shapes Ex.1 + H.W.

21- Simulation of Harmonic analysis (forced vibration) to extract natural frequency & modes shapes Ex.2 + H.W.

22- Simulation of dynamic (unsteady) analysis {$M\ddot{x} + C\dot{x} + Kx = \sum F$} Ex.1 + H.W.

23- Simulation of dynamic (unsteady) analysis {$M\ddot{x} + C\dot{x} + Kx = \sum F$} Ex.2 + H.W.

24- Simulation of Contact analysis Ex.1 + H.W.

25- Simulation of Contact analysis Ex.2 + H.W.

26- Simulation of fluid analysis using fluent Ex.1 + H.W.

27- Simulation of fluid analysis using fluent Ex.2 + H.W.

28- Simulation of fluid analysis using fluent Ex.3 + H.W.

29- Simulation of fluid analysis using CFX (one way interaction FSI) Ex.1 + H.W.

30- Simulation of fluid analysis using CFX (two way interaction FSI) Ex.2 + H.W.

31- Simulation of thermal induced stress Ex.1 + H.W.

32- Simulation of thermal induced stress Ex.2 + H.W.

-8

CAE System

Introduction:

There are many practical engineering problems for which we cannot obtain exact solutions. This inability to obtain an exact solution may be attributed to either the complex nature of governing deferential equations or the difficulties that arise from dealing with the boundary and initial conditions. To deal with such problems, we resort to numerical approximations. In contrast to analytical solutions, which show the exact behavior of a system at any point within the system, numerical solutions approximate exact solutions only at discrete points.

Computer Aided Engineering:

The use of computers to help with all phases of engineering design work. Like computer aided design (CAD), but also involving the construction and analysis of objects, the idea is to use computer processing and interactive computer graphics to enable engineers to create, modify and analyze designs and hence to determine the structural, thermal, flow-field characteristics or other state of a system. CAE programs may use a geometry definition from a CAD program as a starting point, and usually utilize some form of finite element analysis (FEA) as the means to perform the analysis.

Advantages of CAE system:

- **Capable of carrying out different engineering analyses such as, stresses and deformations, buckling, contact analyses, plastic deformations, vibration, heat transfer, fluid flow, magnetic field, coupled field problems, design optimization, etc.**
- **Can work interactively with CAD systems.**
- **Analyses are facilitated through GUI (Graphical User Interface).**
- **Different types of material properties can be included, isotropic, orthotropic, non-linear, etc.**
- **Reduction of time.**
- **Analysis and Simulation can be modified and revised easily.**
- **Graphical presentation of results.**

Disadvantages of CAE system:

- **High cost of CAE software.**
- **Required special and advanced hardware.**
- **Optical fatigue.**
- **High cost of users training and qualification.**

<u>What is ANSYS؟</u>

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solves them all, creating a

comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

Generic Steps to Solving any Problem in ANSYS:

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

1- Build Geometry

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS. In this step we can be drawing the geometry for analysis system that we want to simulate, we can make that in two method the first method by the import the geometry if it drawing in another program and saved on the computer by press the right click on the geometry step and choose import geometry, and the second method by drawing the geometry in design modular program by press double click on the geometry step when the design modular program opened we can draw the required geometry.

2- Define Material Properties

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

3- Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces . After the geometry drawing done, the next step is subdivided this geometry into small size region called element, the element is a closed loop of edges connected at discrete point called nodes.

4- Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions. To solve any analysis system we must defined the load on the boundary; this load must be coinciding to the nature of case to ensure the correct solution, and this boundary condition different depending on the analysis system.

5- Obtain Solution

This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved. After the setup complete, the next

step is solution the case. This solution is representing by the solution the variable that we want to get it which setting in the setup step and this solution different from case to another depended on the nature of the case.

6- Present the Results

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots. When the solution done we can view the solution result and this result can be represent in Graphics and Animations, Plots and Report. From Graphics and Animations can get on contour and vector in any choosing location.

Tutorial 1:

Introduction to ANSYS

Introduction:

This Tutorial will use a readymade file to speed up the learning process for the student. This file is provided in Parasolid format. The intention of this tutorial is to get the student to run a straight forward simulation. By the end of this tutorial a check list for the required procedure can be formulated by the student. ANSYS as a software is made to be user-friendly and simplified as much as possible with lots of interface options to keep the user as much as possible from the hectic side of programming and debugging process.

Why is it that such a simple model is used?

During this tutorial a simple geometry is used, the objective of that is that the student masters the steps to get to run a simple simulation, once that's done the student can model any kind of geometry he sees necessary for his studied case.

Step1: Launch ANSYS ,by going to the start-up menu and double clicking on workbench file in the ANSYS 13.0 folder.

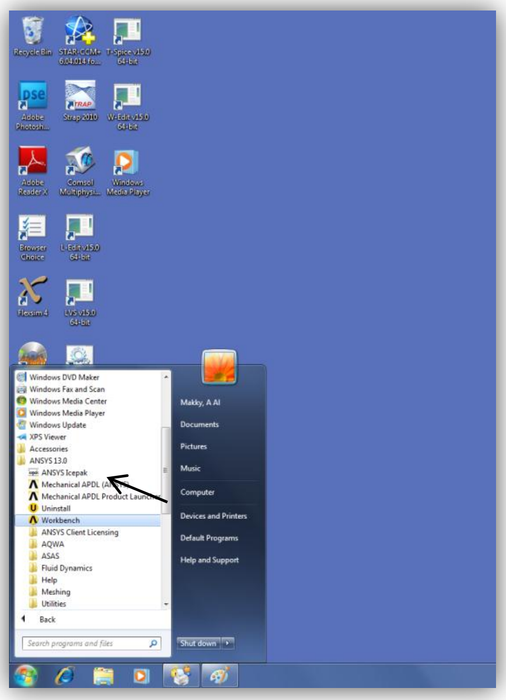


Figure1: A reminder that not all lab machines have the ANSYS software installed on them.

Step2: Once the program is launched it should look like as shown below. Go to Analysis Systems Fluid Flow (CFX) and double click.

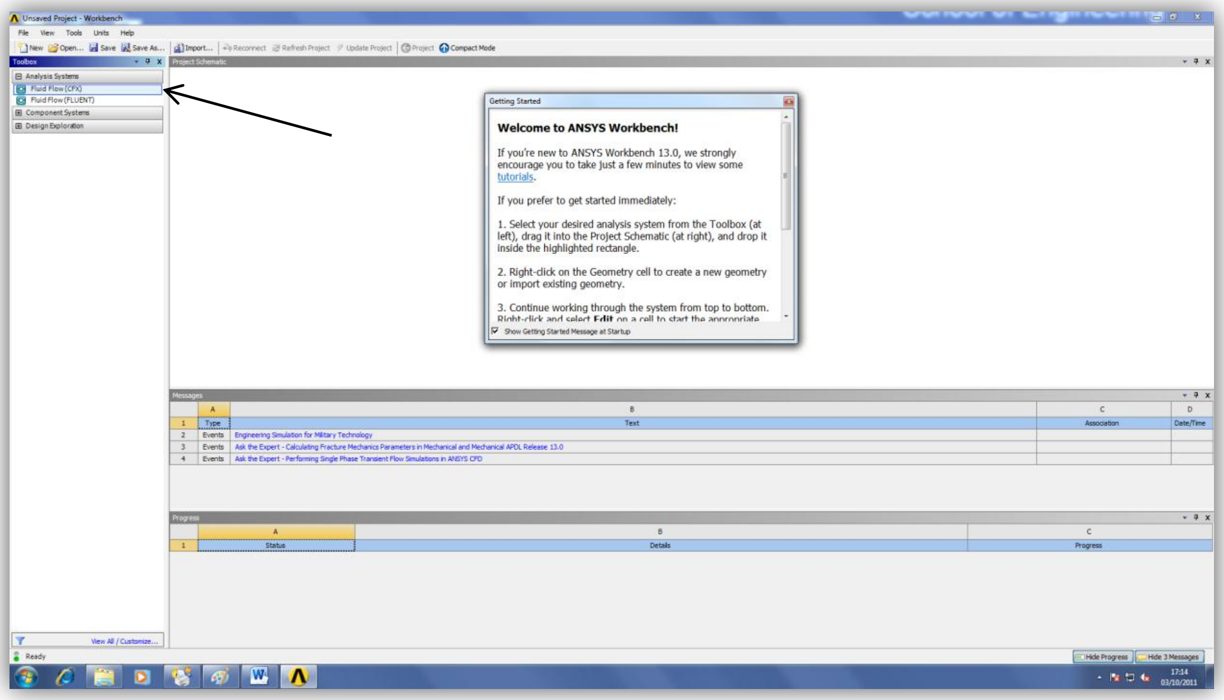


Figure2: You might have to wait a bit till ANSYS gets running, the student is encouraged to use the provided help with the software, it has lots of useful hints here and there.

Step3: Next Double click on the Geometry. This stage is for getting the required geometry read into the software, note that there is a blue question mark icon beside the geometry text. Looking at the bottom of the window you will see two windows one having the title of **Messages**, this title confirms that the imported geometry has no problems with it, the next window has the title **Progress** and that is necessary to prove that state of the progress and if there is a problem it will state the problem.

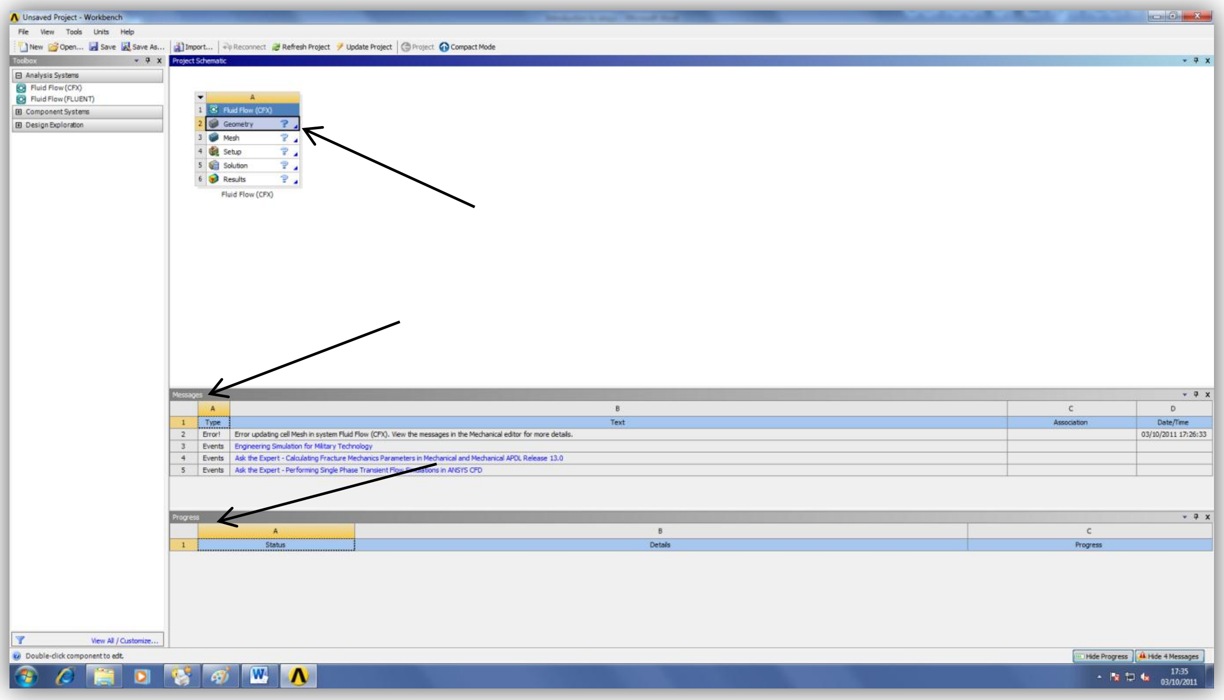


Figure3: At the moment the illustration are a bit simplified for the user and will get complex with time.

Step4: Once ANSYS Workbench window is active you will get a window asking to specify working units for the model dimension chose meters and press ok. For the user this step might seem secondary in importance but as a matter of fact it's of great importance, because at later stages you will have to specify the box size (discrete element dimension). Box size dimension leads to finer mesh, the finer the used mesh is the more accurate is the captured data. The captured data term refers to the fluid flow structures.

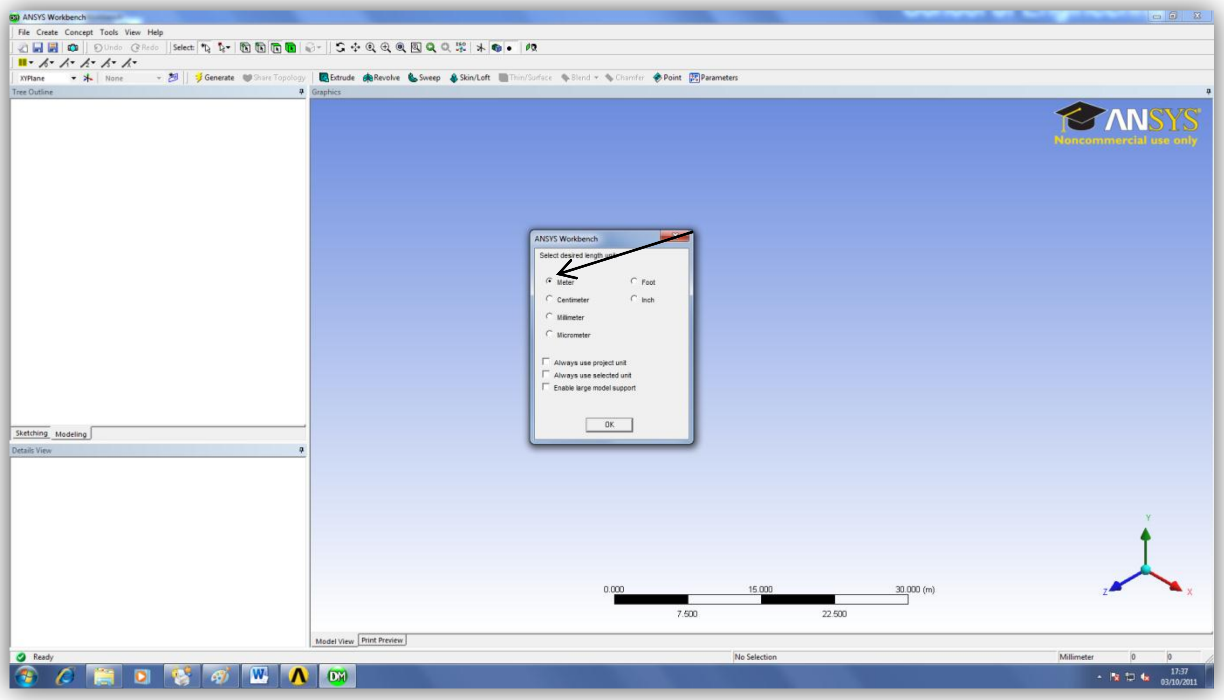


Figure4: Depending on your studied case the selection of serial or parallel is taken, also depending on the hardware provided in the computer lab dual core or quad core etc.

Step5: Go to file and choose Import External Geometry File…. .

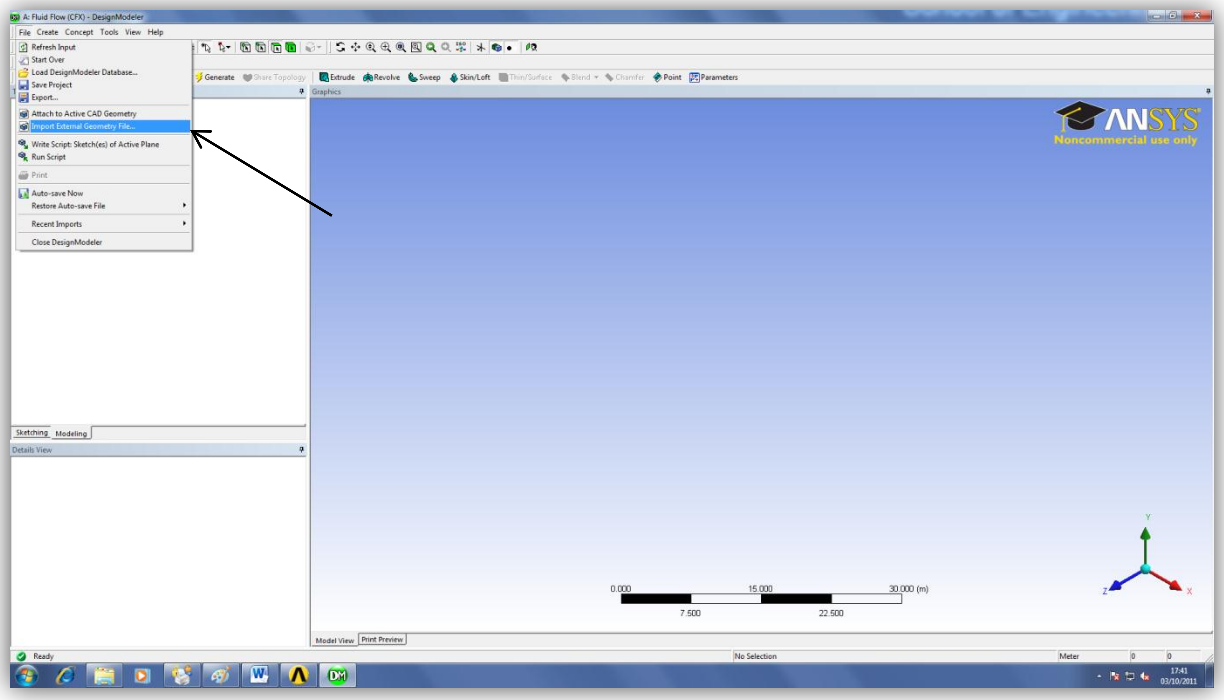


Figure 5: You can model your geometry using the sketching tools provided with DesignModeler.

Step 6: A window having a title open will be visible to the user, choose File type Parasolid(*x_t;*xmt_txt;*x_b;*xmt_bin) then go to the folder that has the required file .

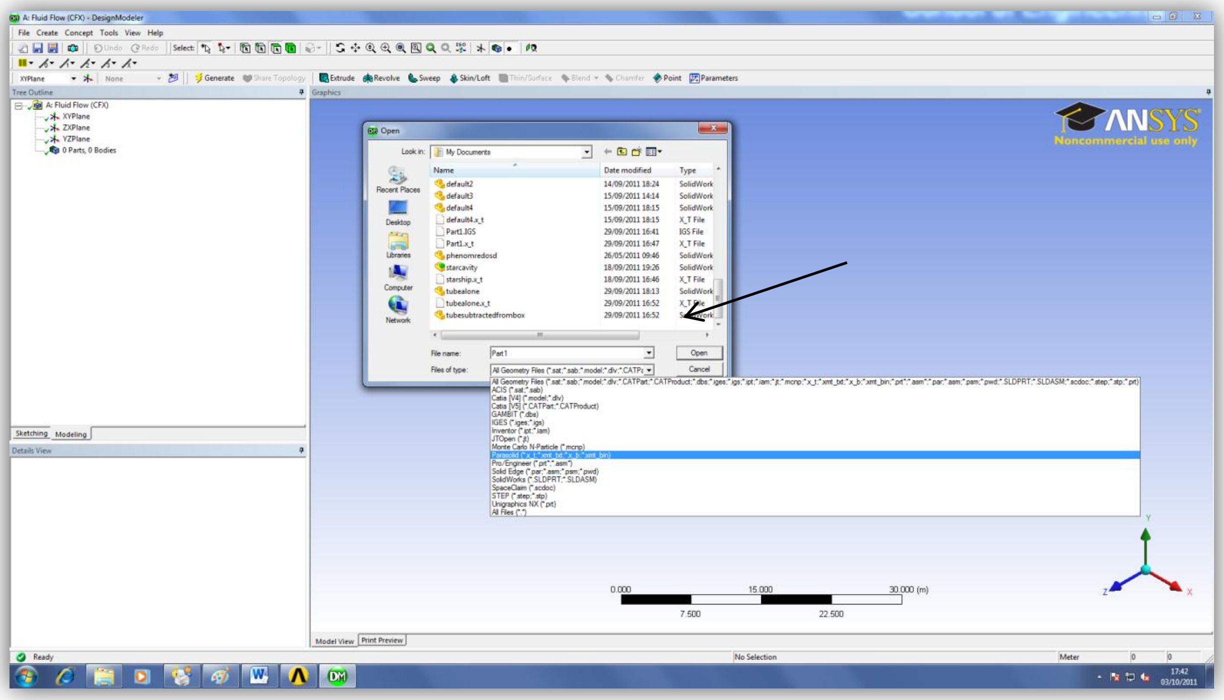


Figure 6: There are lots of software that are used to generate meshes, depending on the software used the file extension text would be, in our case we are using SolidWorks to generate the mesh and then exporting it in Parasolid format. A question comes to the mind of the student why do I have to specify the file extension. The answer is that each mesh generation software has its own structure in its generated data sets. A simple example:

Software 1:

N	x	y	z
1	1*dx	1*dy	1*dz
2	2*dx	2*dy	2*dz
3	3*dx	3*dy	3*dz

Software 2:

N	1	2	3
x	1*dx	2*dx	3*dx
y	3*dy	3*dy	3*dy
z	3*dz	3*dz	3*dz

Step7: Looking at the DesigModeler window, we can't see the imported geometry yet, what is required next is to press on the generate icon that is represented by a yellow thunder icon.

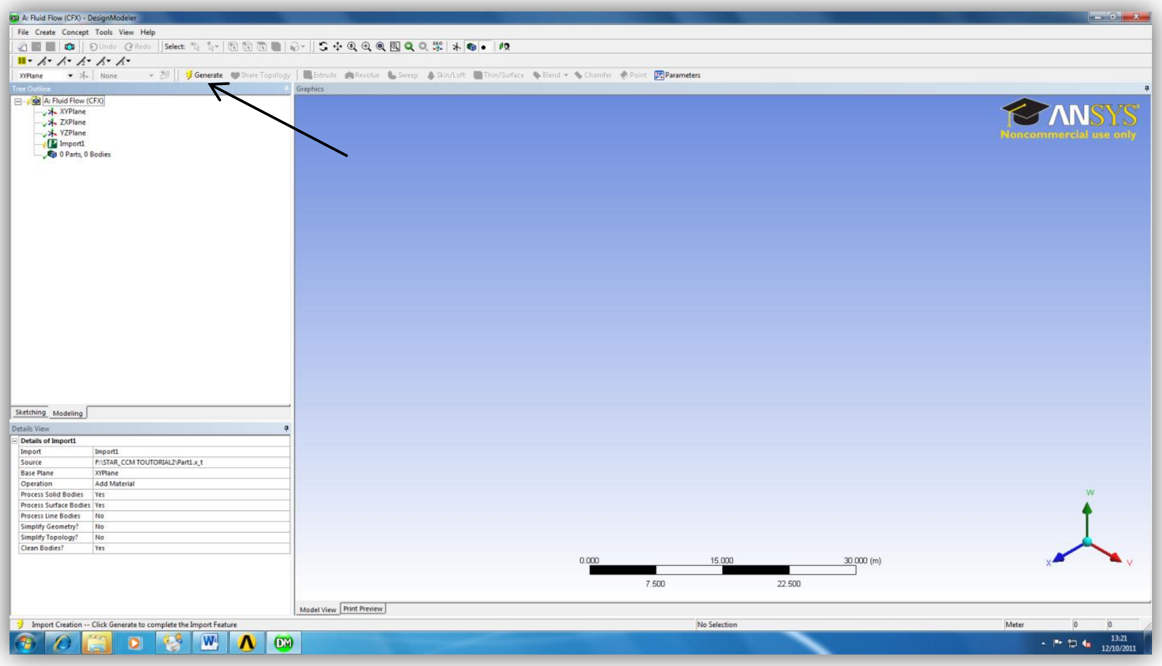


Figure6: The DesignModeler will read in the imported data file, and will construct the required mesh.**Step7:** The imported Geometry Domain should look something like this, still that doesn't give any hints to the user, relating to the inner structure of the domain.

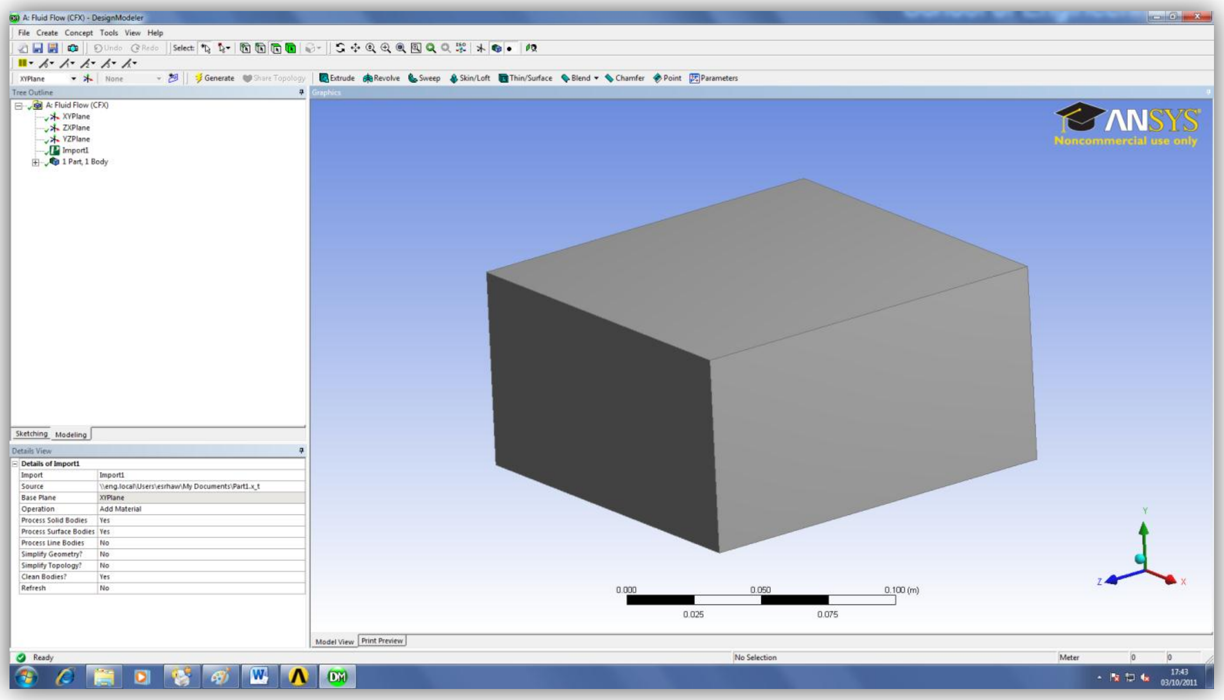


Figure7: The geometry domain is viewed in the shaded exterior style.

Step8: go to view and chose wireframe.

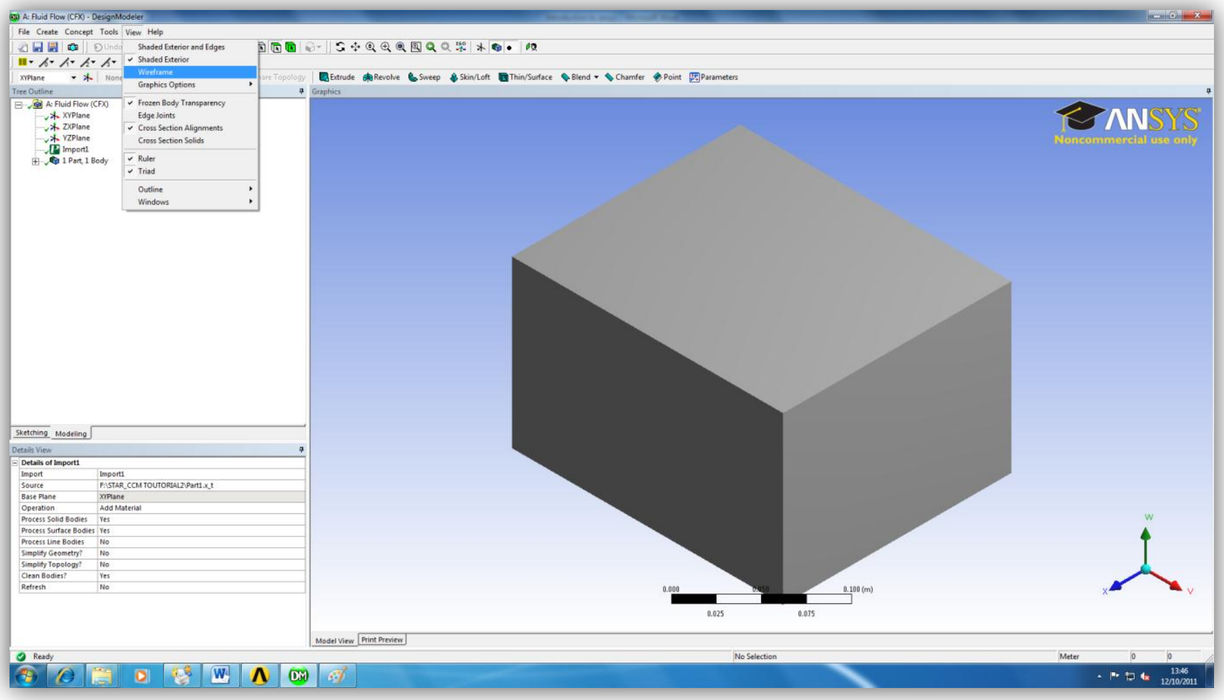
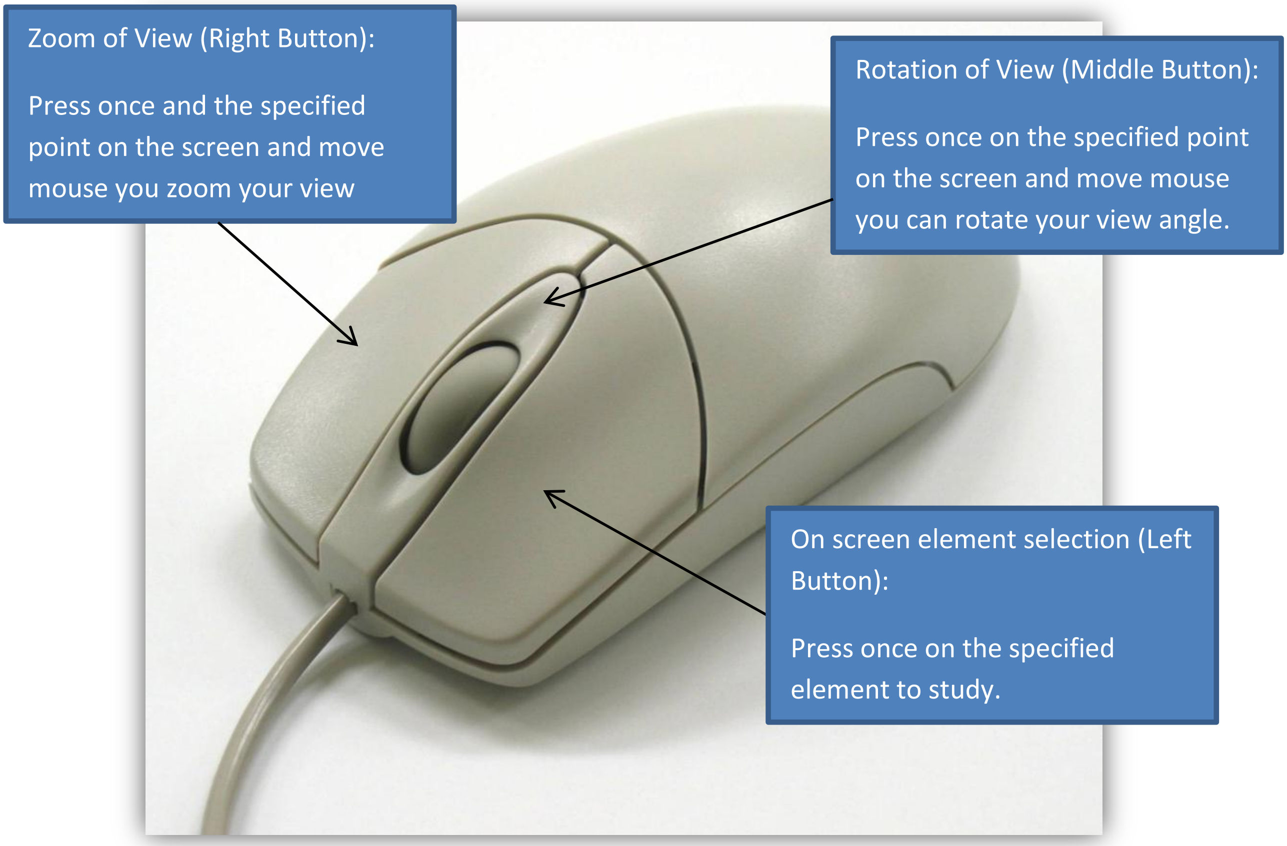


Figure 8: This step is necessary to view the inner structure of the domain.


Zoom of View (Right Button):
Press once and the specified point on the screen and move mouse you zoom your view
Rotation of View (Middle Button):
Press once on the specified point on the screen and move mouse you can rotate your view angle.
On screen element selection (Left Button):
Press once on the specified element to study.

Step 9a: Once the student gets to this stage, that means he has finished from the DesignModeler and has to proceed to the Meshing part.

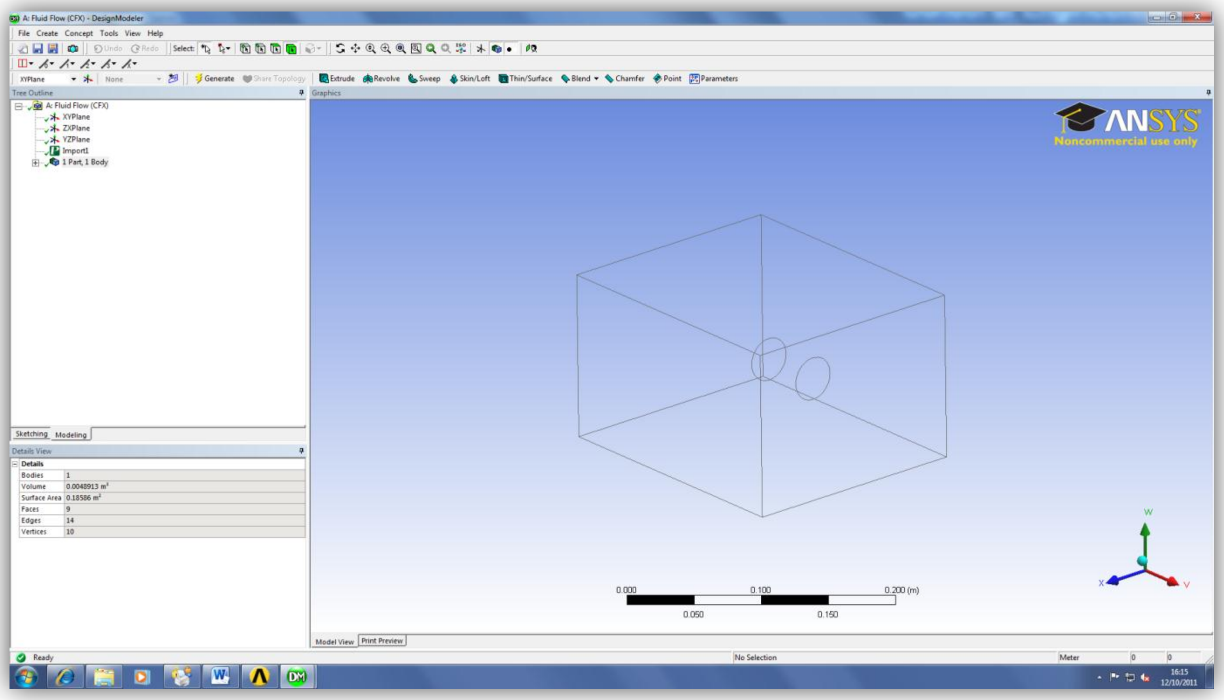


Figure 9a: Rotate the view and check that the Geometry satisfies the design requirements.

Step 9b: Go to the workbench and check that there is a green tick sign beside the Geometry and then double click on the Mesh Icon.

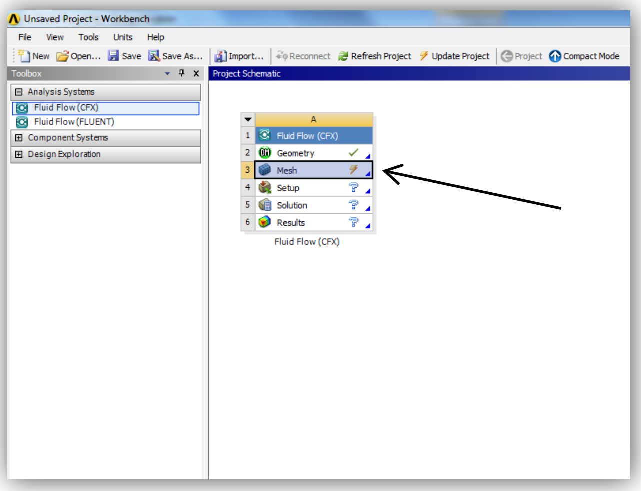


Figure 9b: Congratulations you have finished from DesignModeler and now have started with the Meshing part.

Step 10a: The Meshing part of the project has started, notice that beside the Mesh there is a yellow thunder icon.

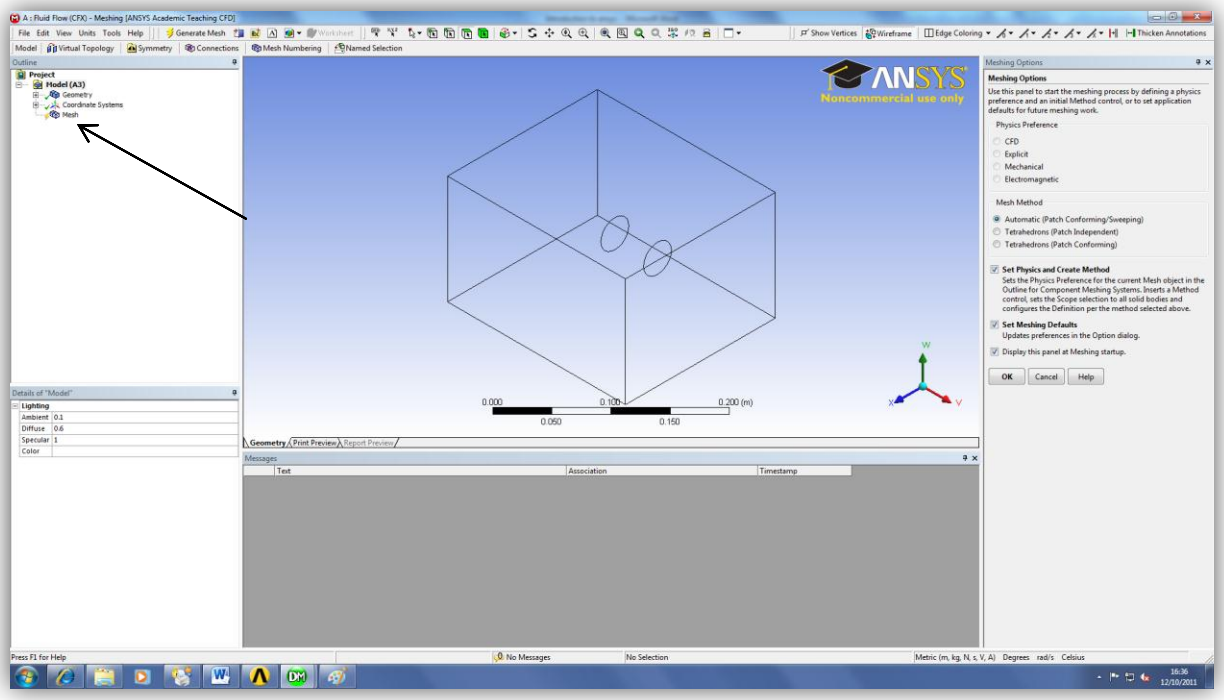


Figure 10a: The scale shown at the bottom helps you make the right decision on the box sizing, sothat we can see that the largest value on the scale is 0.200(m) which means we have to choose a value less than 0.050(m).

Step 10b: right click on Mesh and chose Insert and then chose Method.

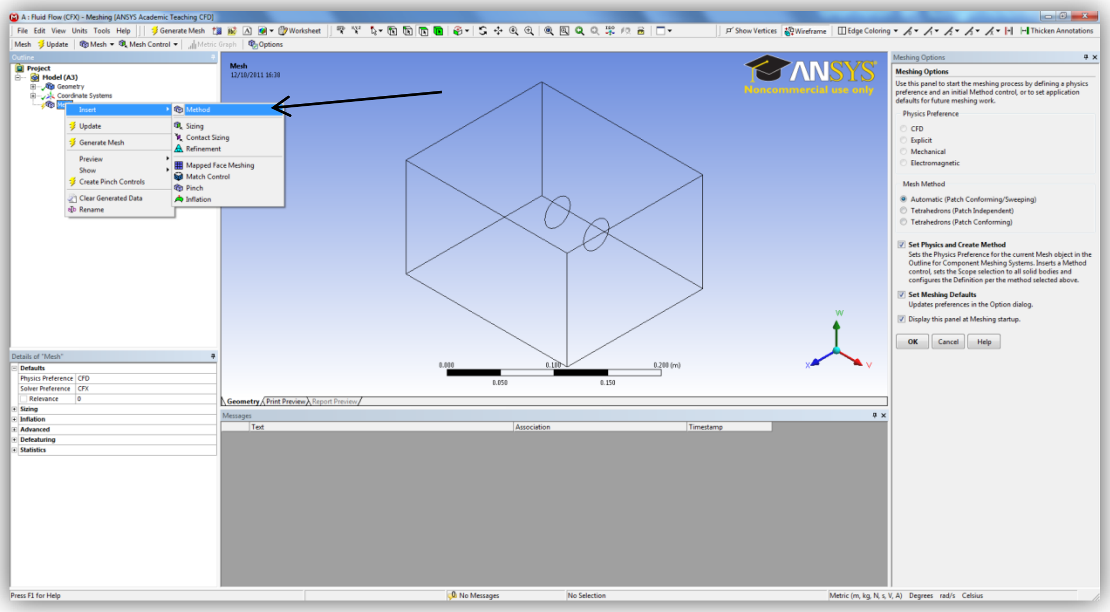


Figure 10b: at this stage we come to the point where we have to choose what kind of mesh are we going to use wither regular or irregular or etc.

Step 10c: click on the positive sign beside the Mesh you should get a tree sub branch have automatic Method using the left button click on the grey box domain, as a result it should by highlighted in green, then you see that the geometry text is highlighted in blue press the apply.

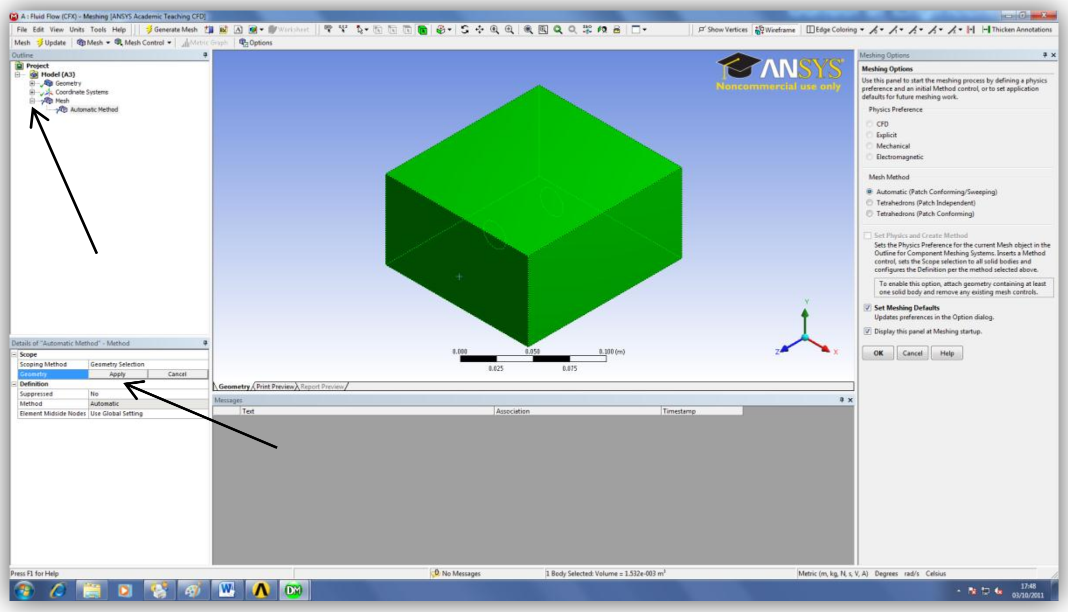


Figure 10c: choose the parallel option in the projection mode, which will come handy later on, when you want to use the measure command or choosing the appropriate slice plane for your study.

Step 11: go to method and choose Tetrahedrons.

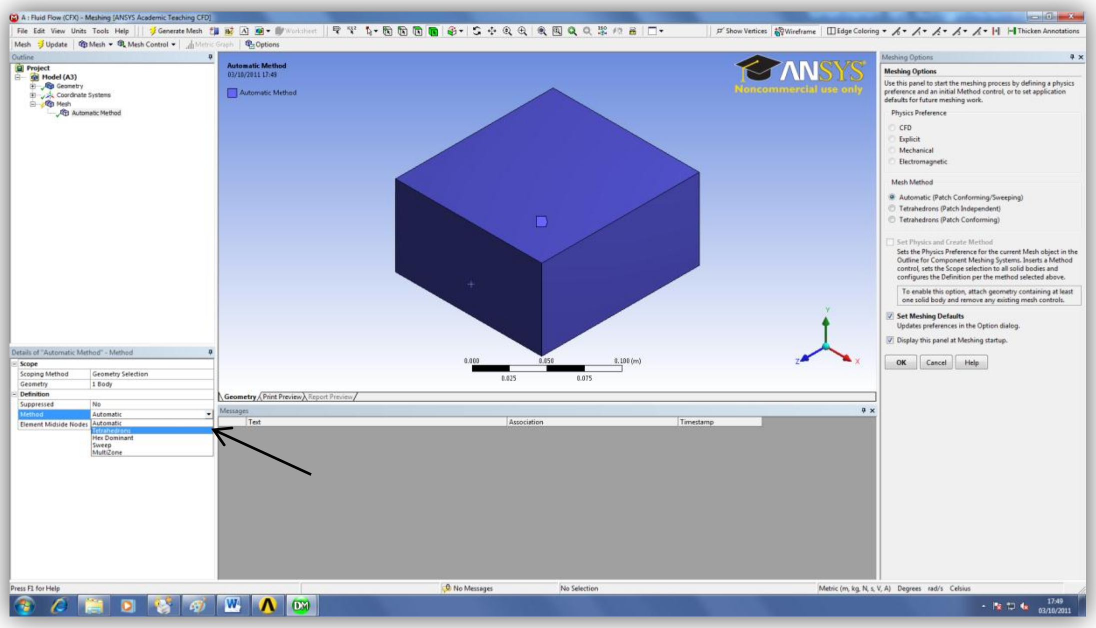


Figure 11: This prepares the view for later wanted operations.

Step 12: Go to algorithms and choose Patch Independent.

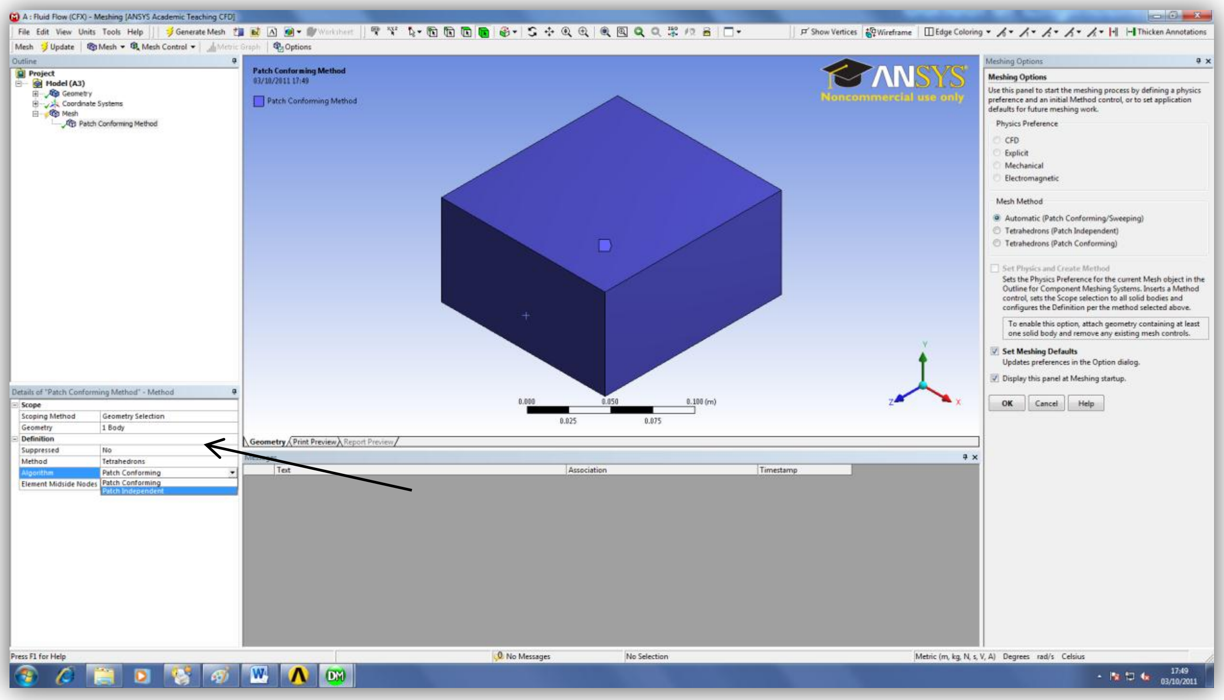


Figure12: Now that you have specified the mesh properties, you can proceed to the next step .

Step13: press the Update icon and then press on the Generate Mesh icon.

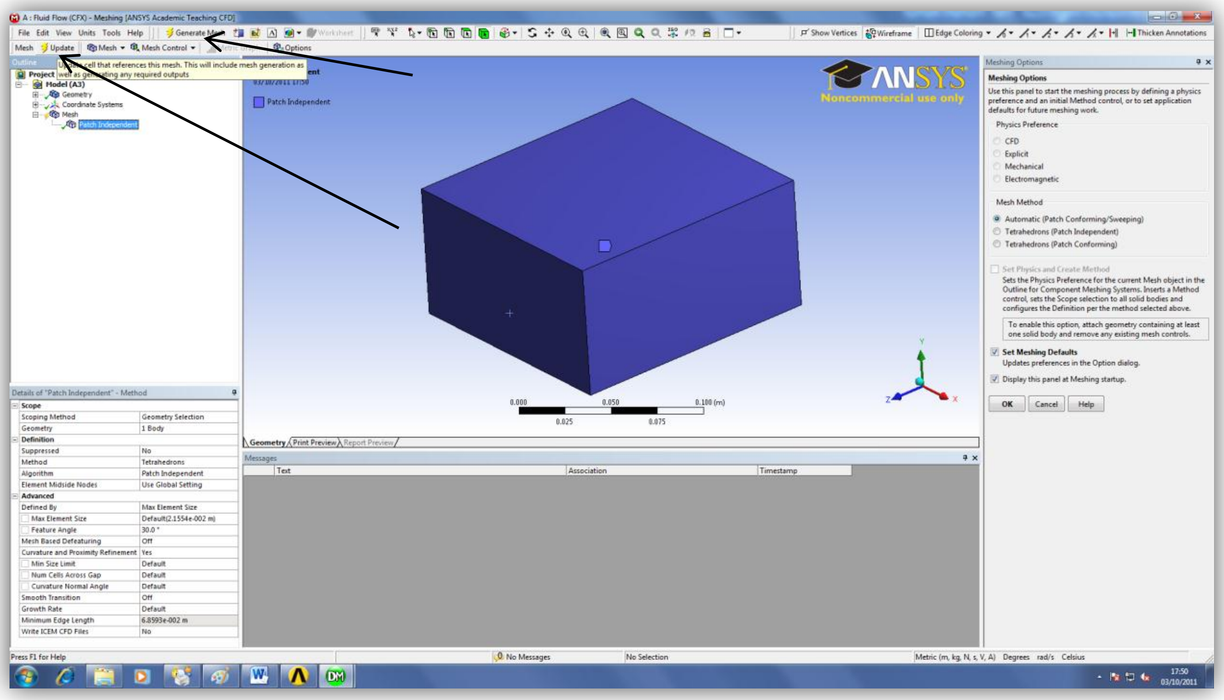


Figure13: For our case we will want to now the dimensions of the inflow section of the pipe.

Step14: click on mesh, now it's visible to the user the generated mesh.

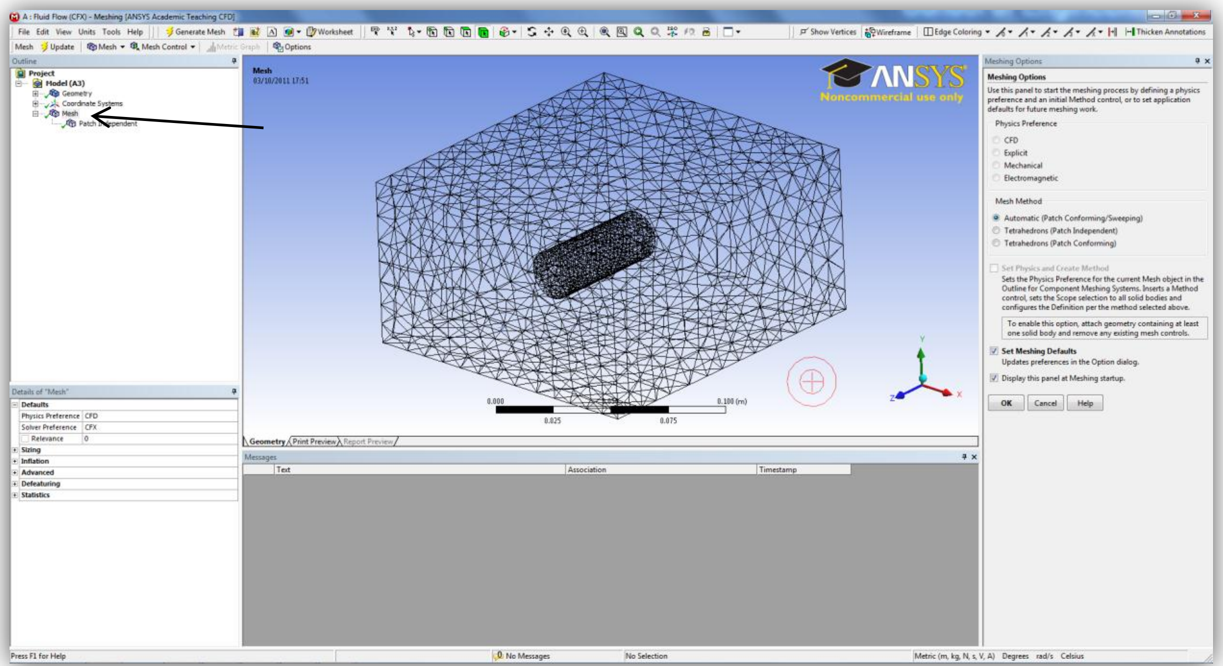


Figure 14: Click on the middle button to rotate the view to inspect your mesh.

Step15: Go to work bench, you will see there is a green tick beside the mesh congratulations you can now proceed to the setup.

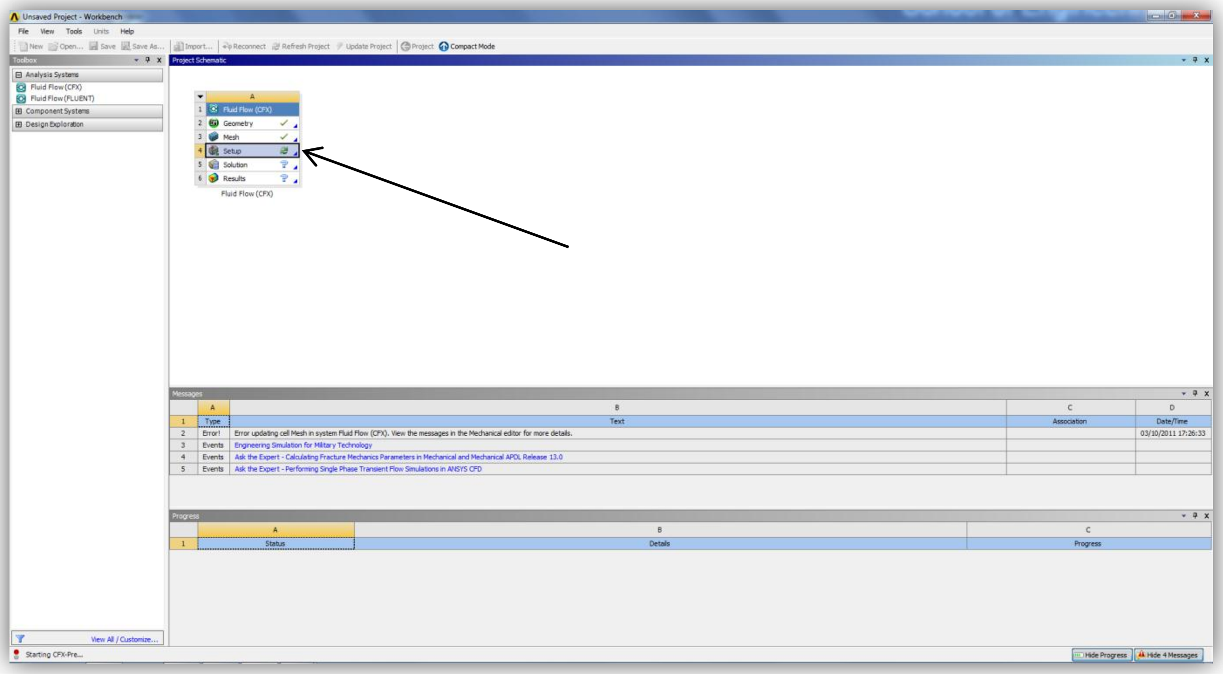


Figure15: Check the messages window if there are any errors you will have to go back in steps and check where you went wrong.