

- - [Print](#)
 - [Workbench](#)
 -
 - [Community](#)
- ◻
- [Log in](#)
- - [Library](#)
 - [Challenges](#)
 - [Groups Beta](#)
 - [Questions](#)
 - [Tutorials](#)
 - [Engineers](#)
- - - [Blog](#)
 - - [Log in](#)
- Other
 - [Help Center](#)
 - [Workbench overview](#)
 - [About us](#)

Learn from thousands of free Tutorials

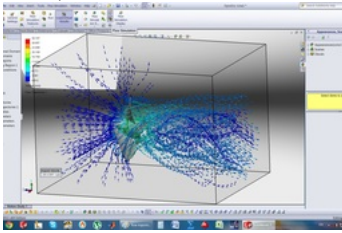
[Join the Community](#)

Tutorial how to do a flow simulation to a rotating propeller?



[Andreas Gkertsos](#)

19 Jan, 2012 08:15 PM

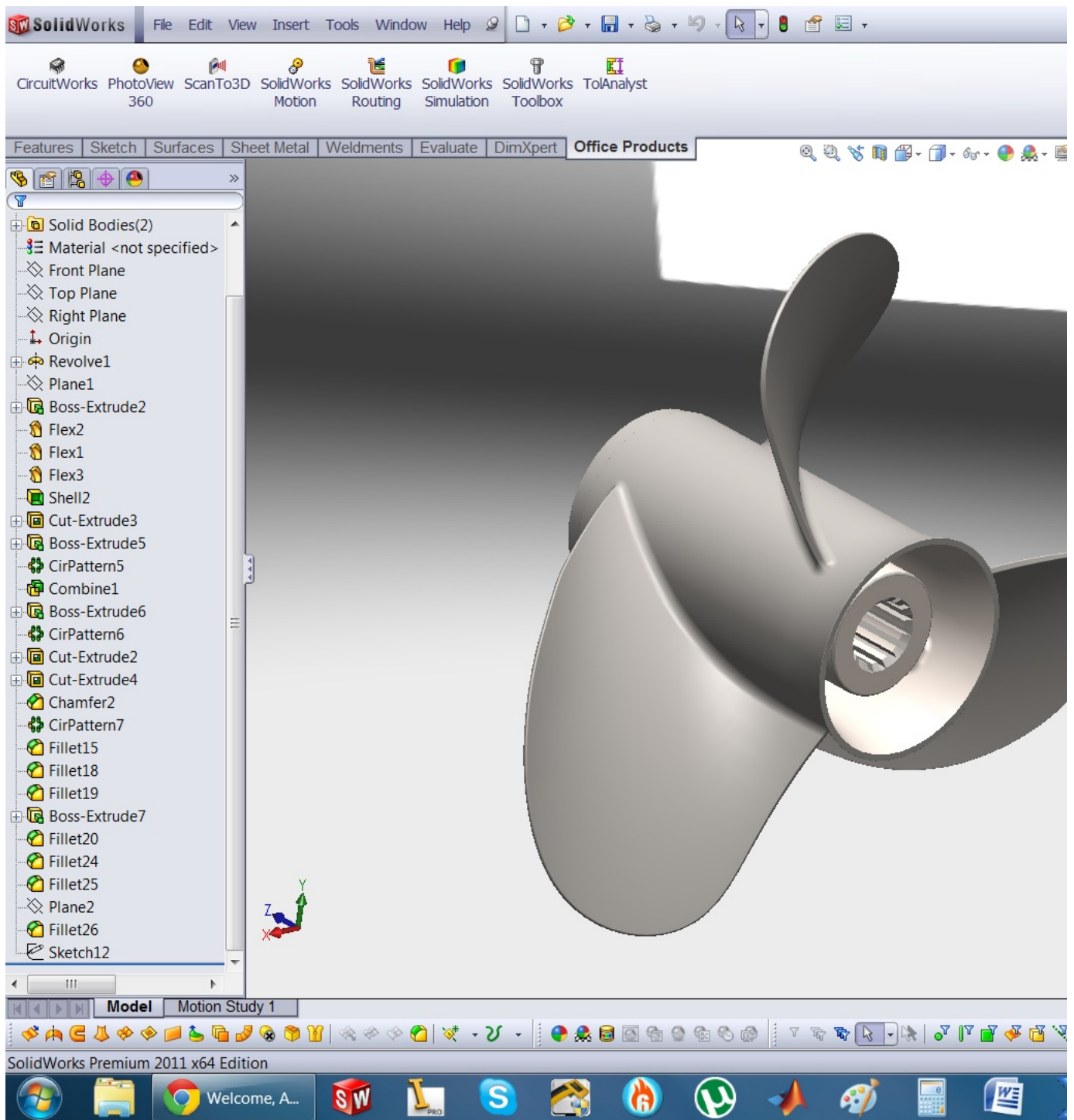


These are the steps to do a flow analysis...

1. Step 1:

First of all we pick a part that will be rotated... (i am using a part that i have designed)

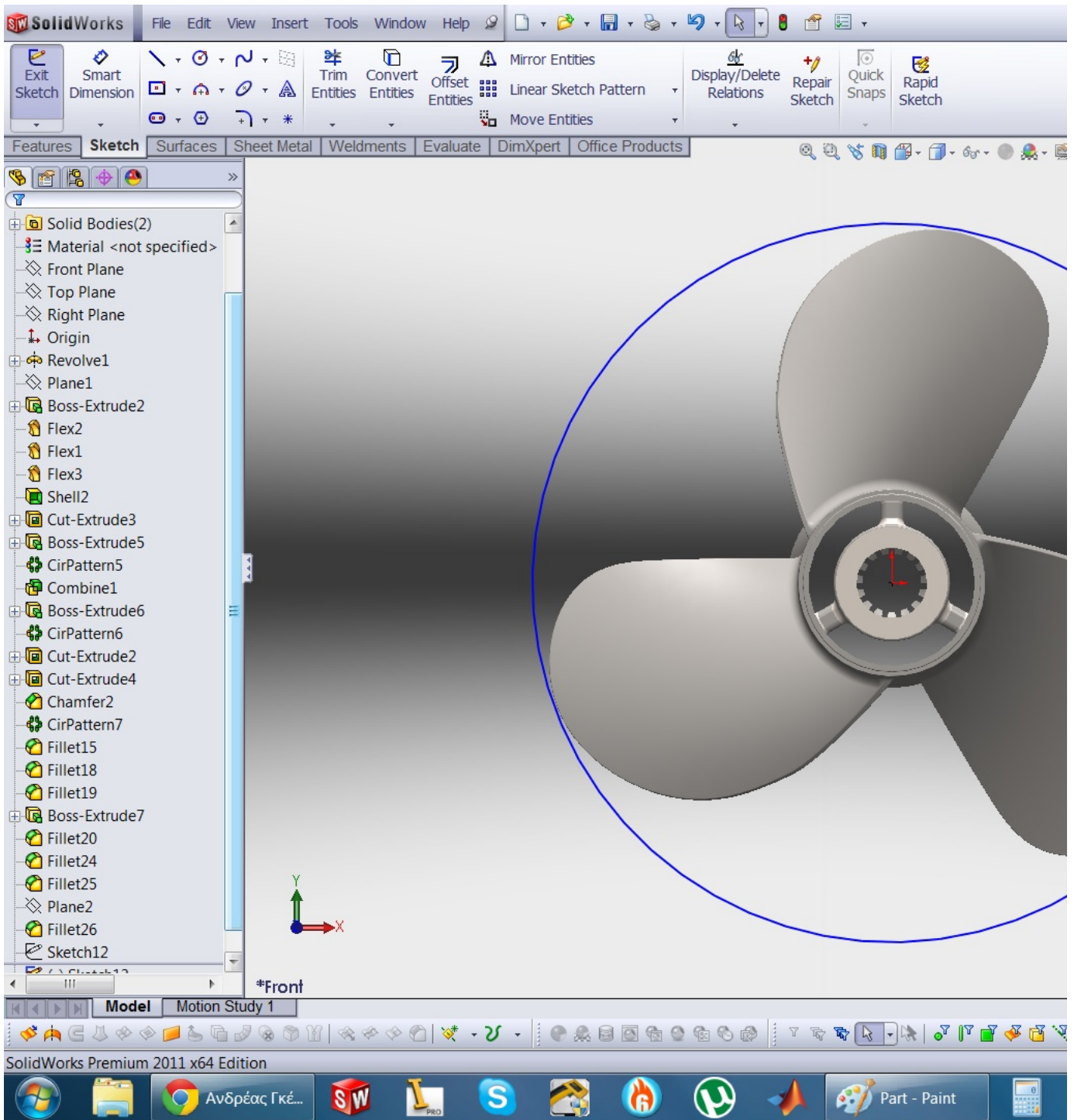




2. Step 2:

then we go to a plane, and we create a cylinder that contains the part we want to rotate...

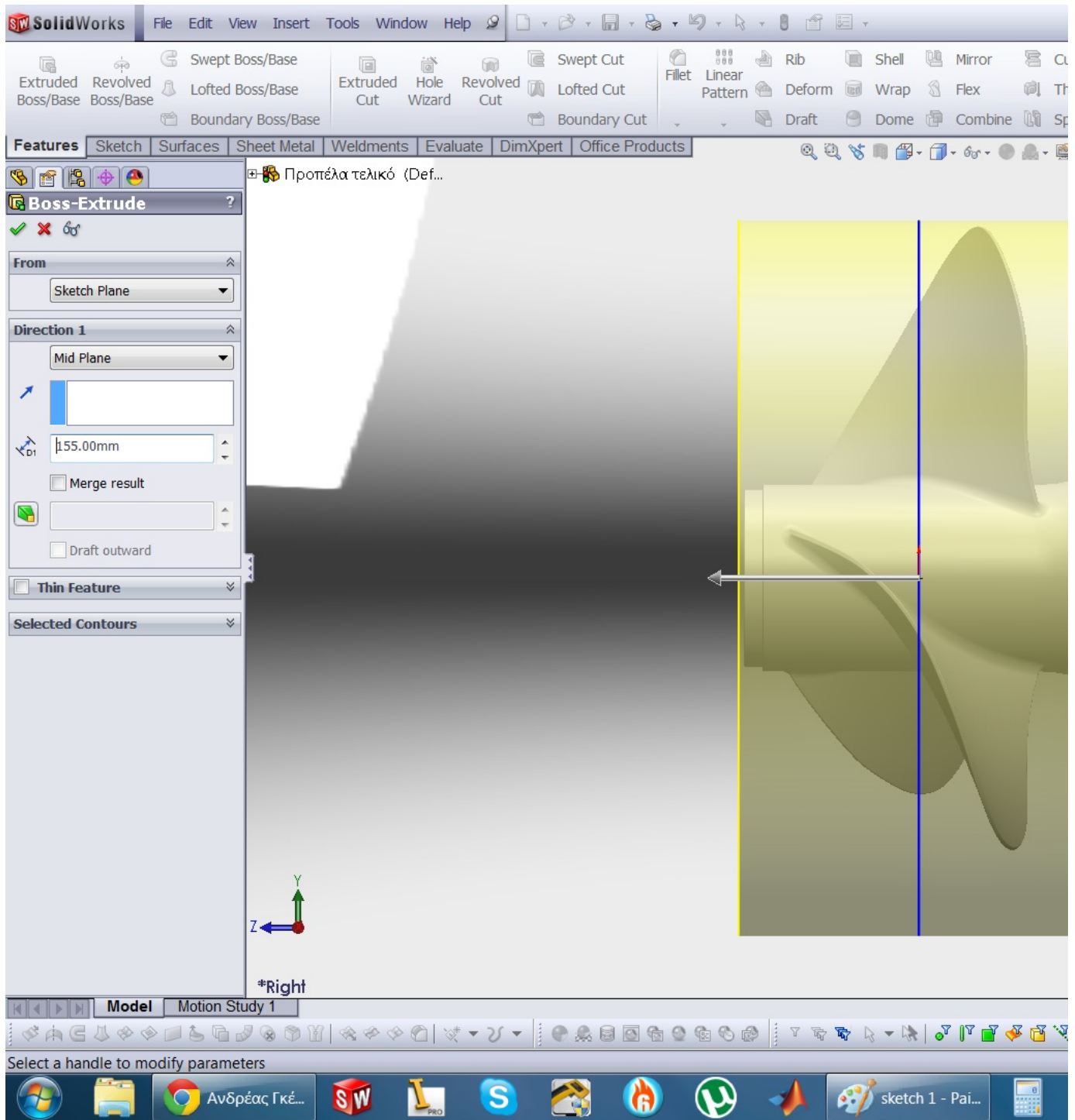




3. Step 3:

And we extrude it WITHOUT merging result... This is the volume that will be rotated...

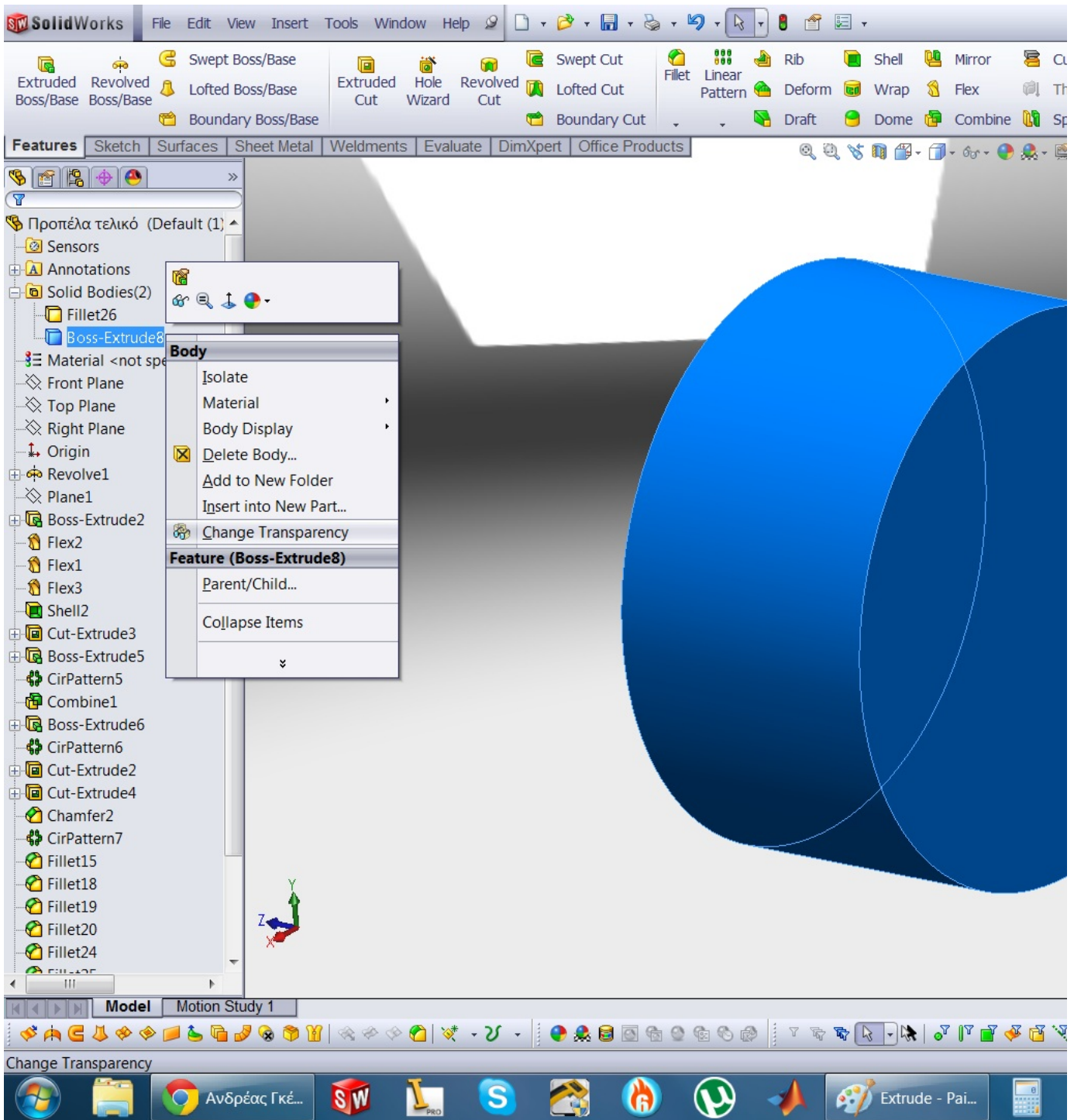




4. Step 4:

then we click change transparency in order to see our part in it...

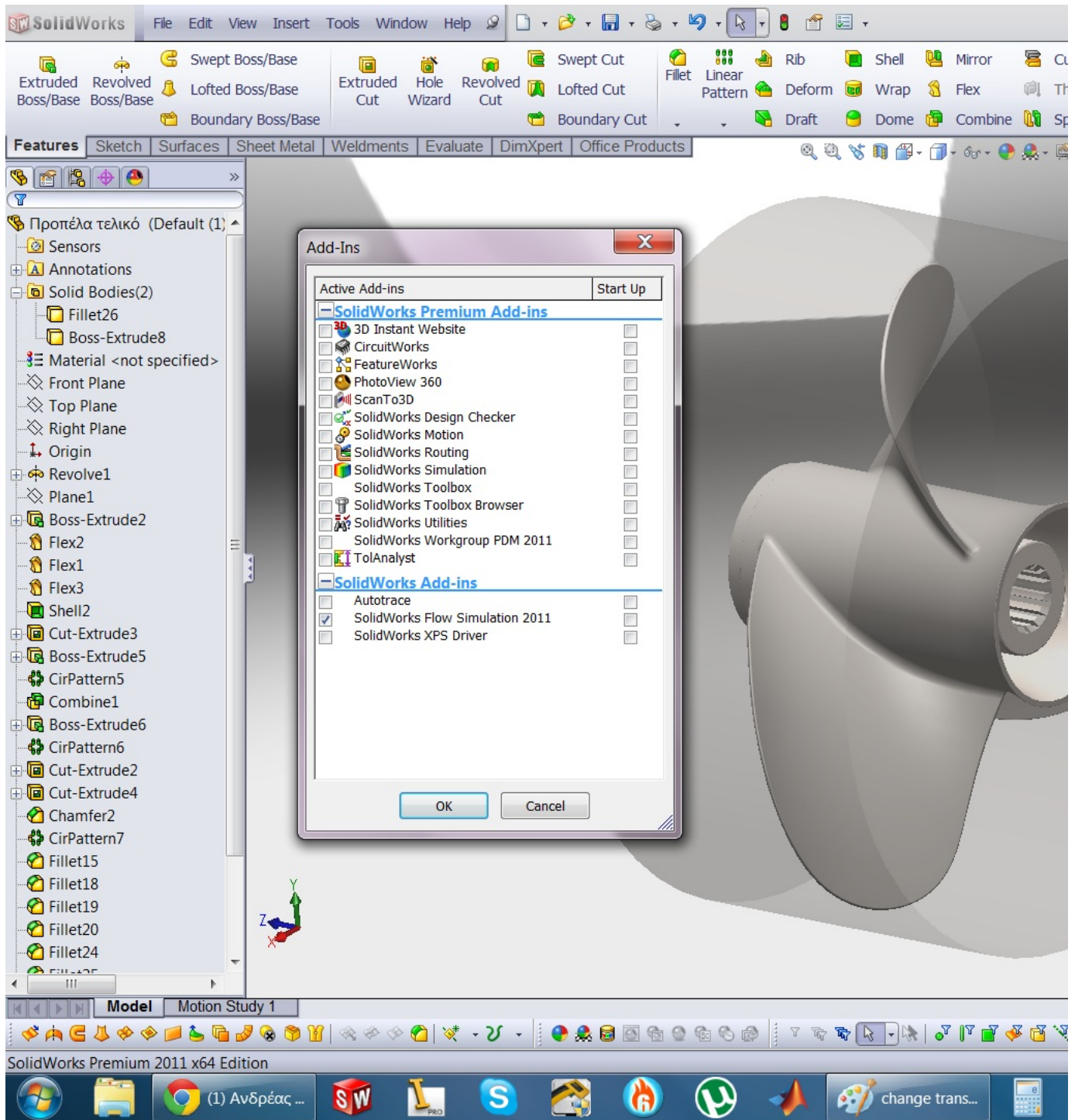




5. Step 5:

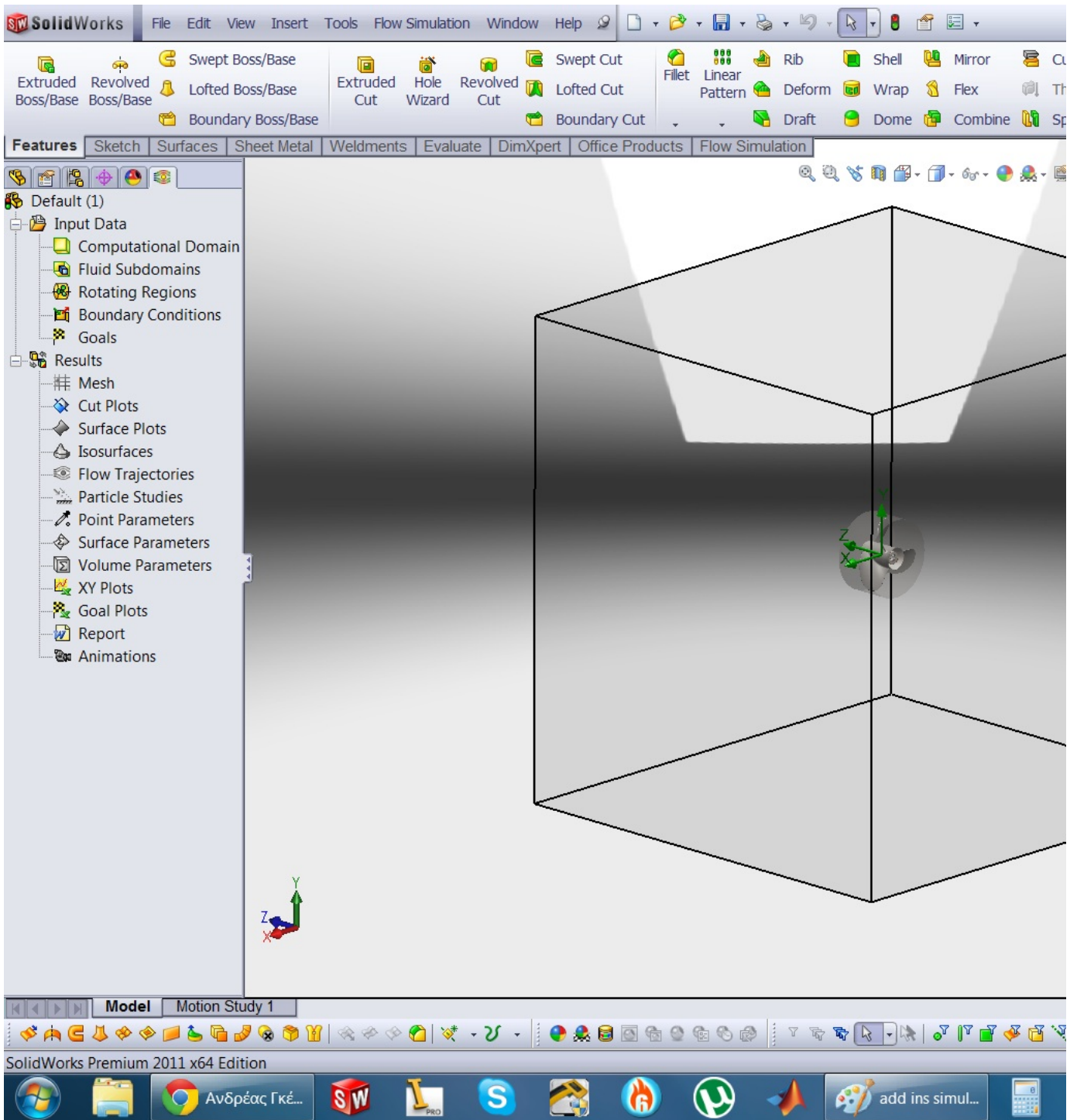
Next we enable the flow simulation command in Tools->Add-ins->Flow simulation... Click ok





6. Step 6:

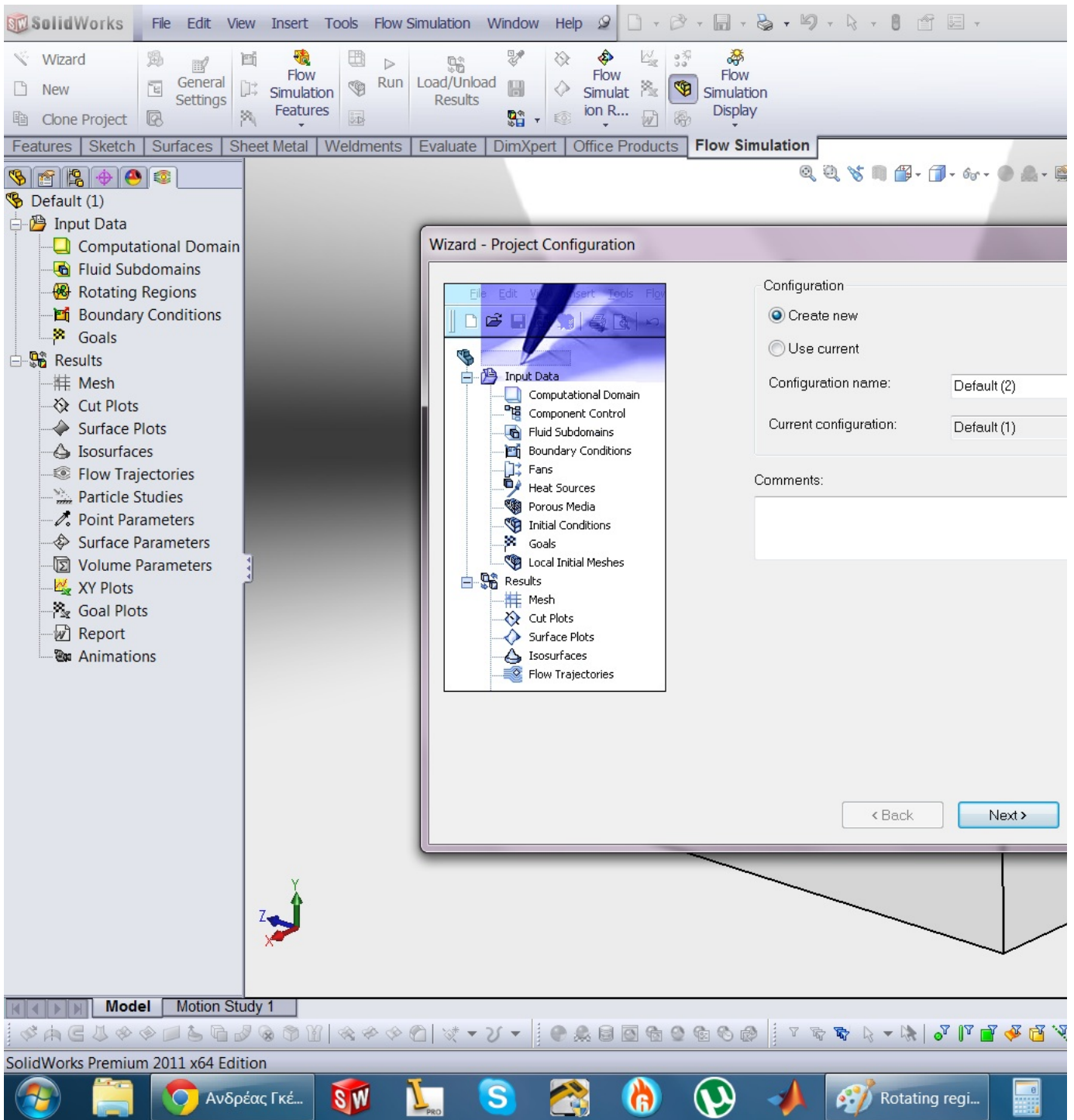
then we open the flow simulation menu



7. Step 7:

We click on wizard in order to create a new study....

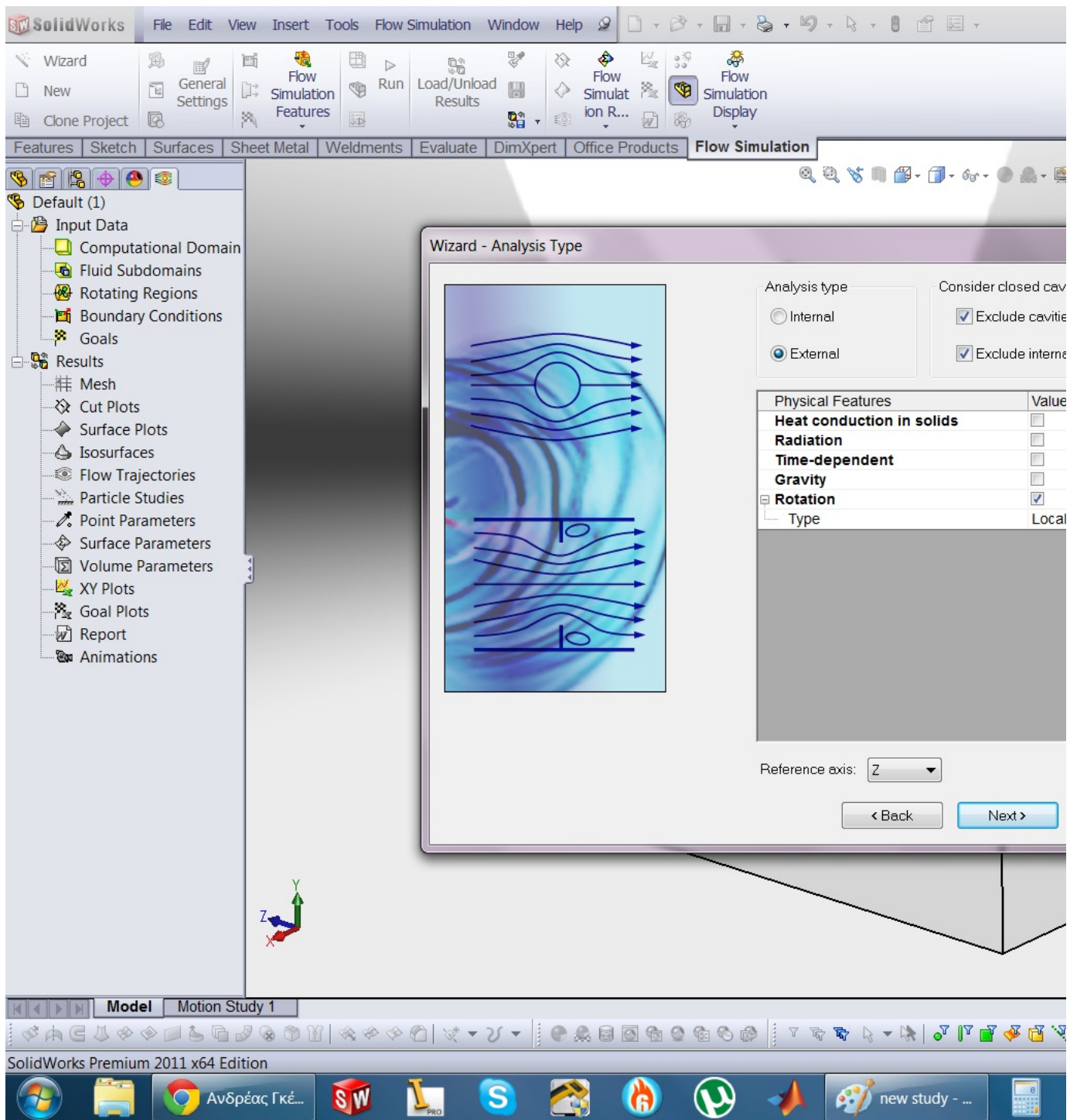




8. Step 8:

we click on the arrows on the right side and click on analysis type... this type indicates that we are not going to give fluid boundaries (speed pressure etc) but we are gonna calculate them...

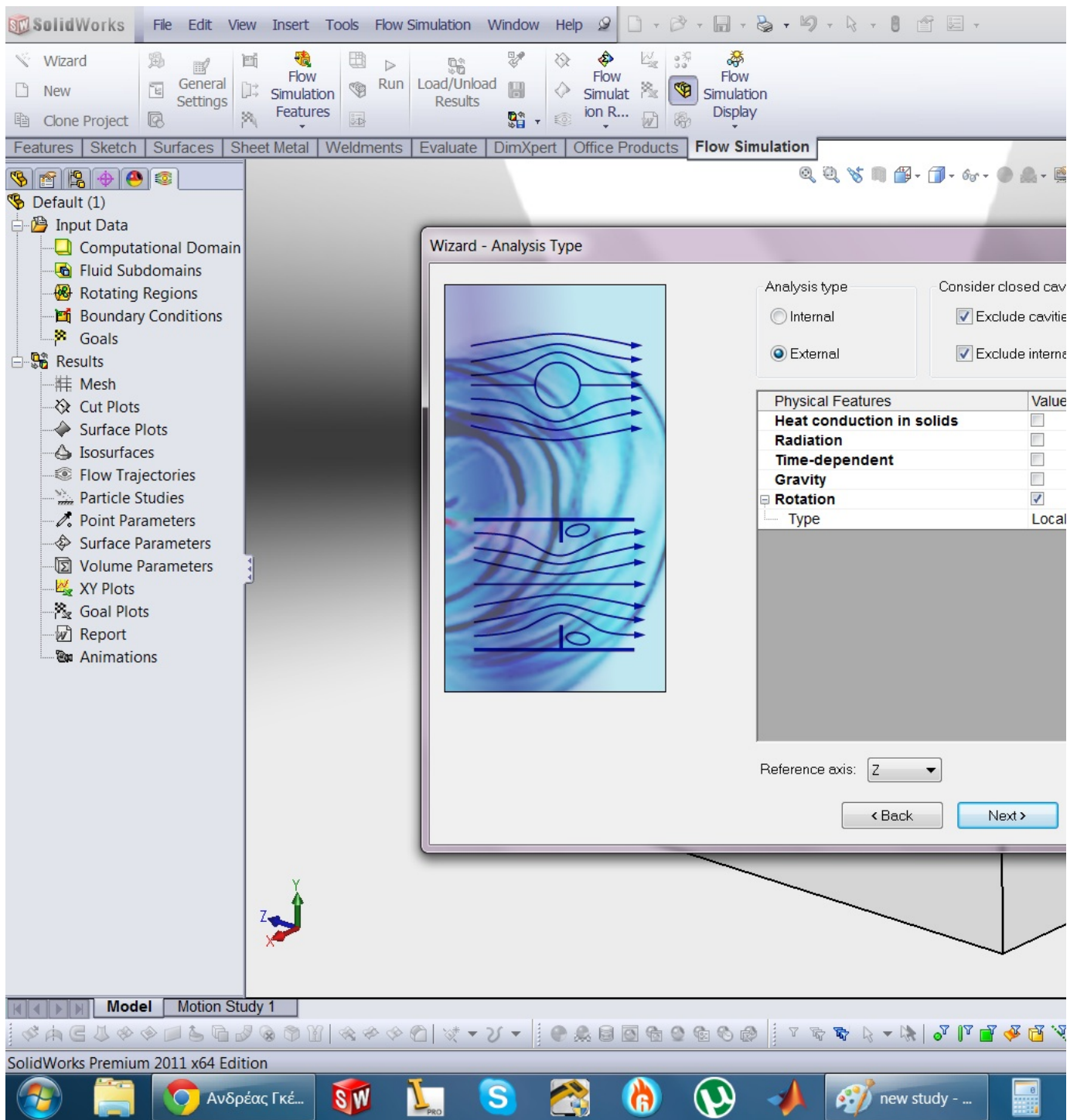




9. Step 9:

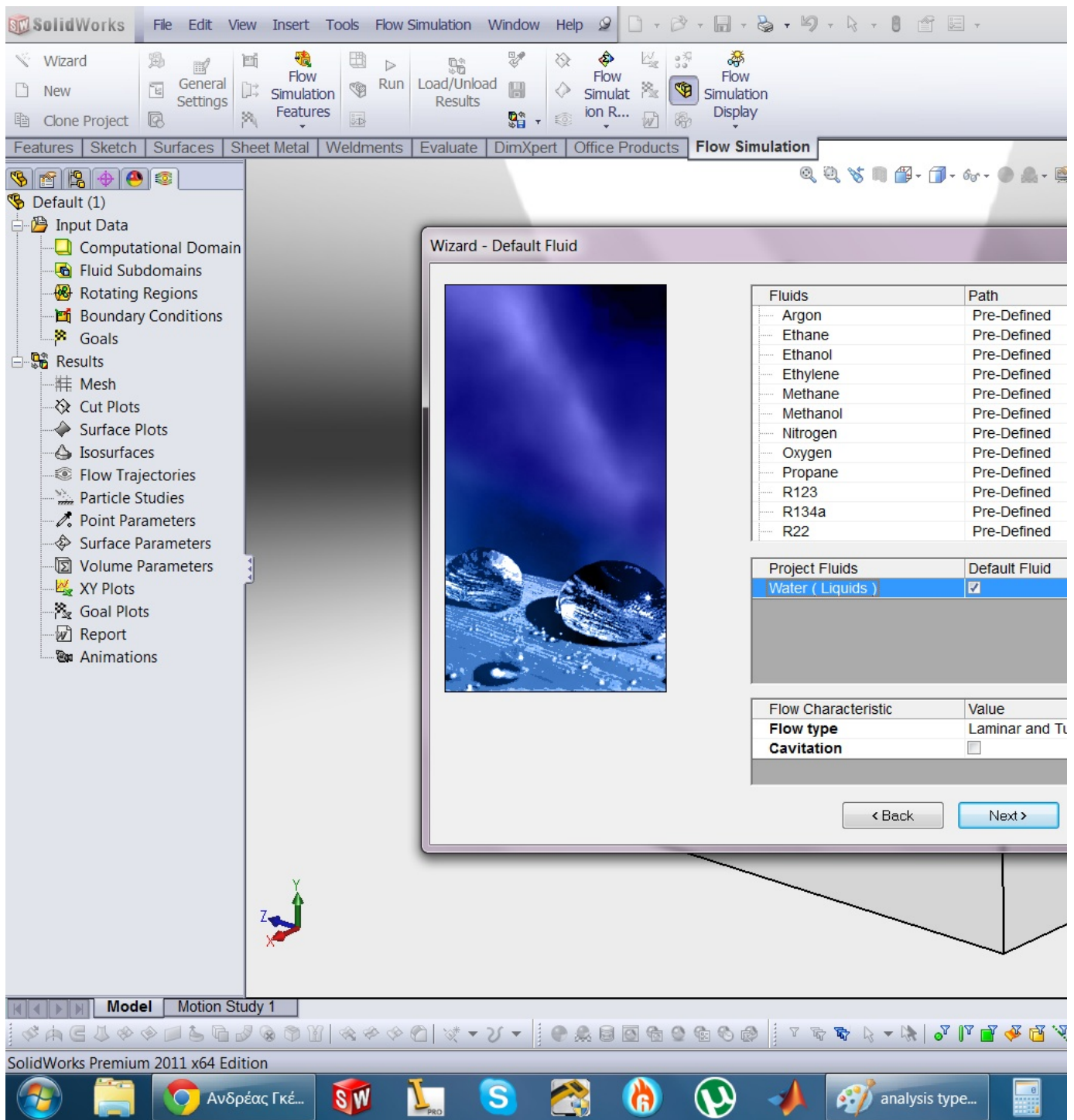
We choose external flow and we check both boxes on the right side... we click on the rotation analysis... (our propeller will be rotating)





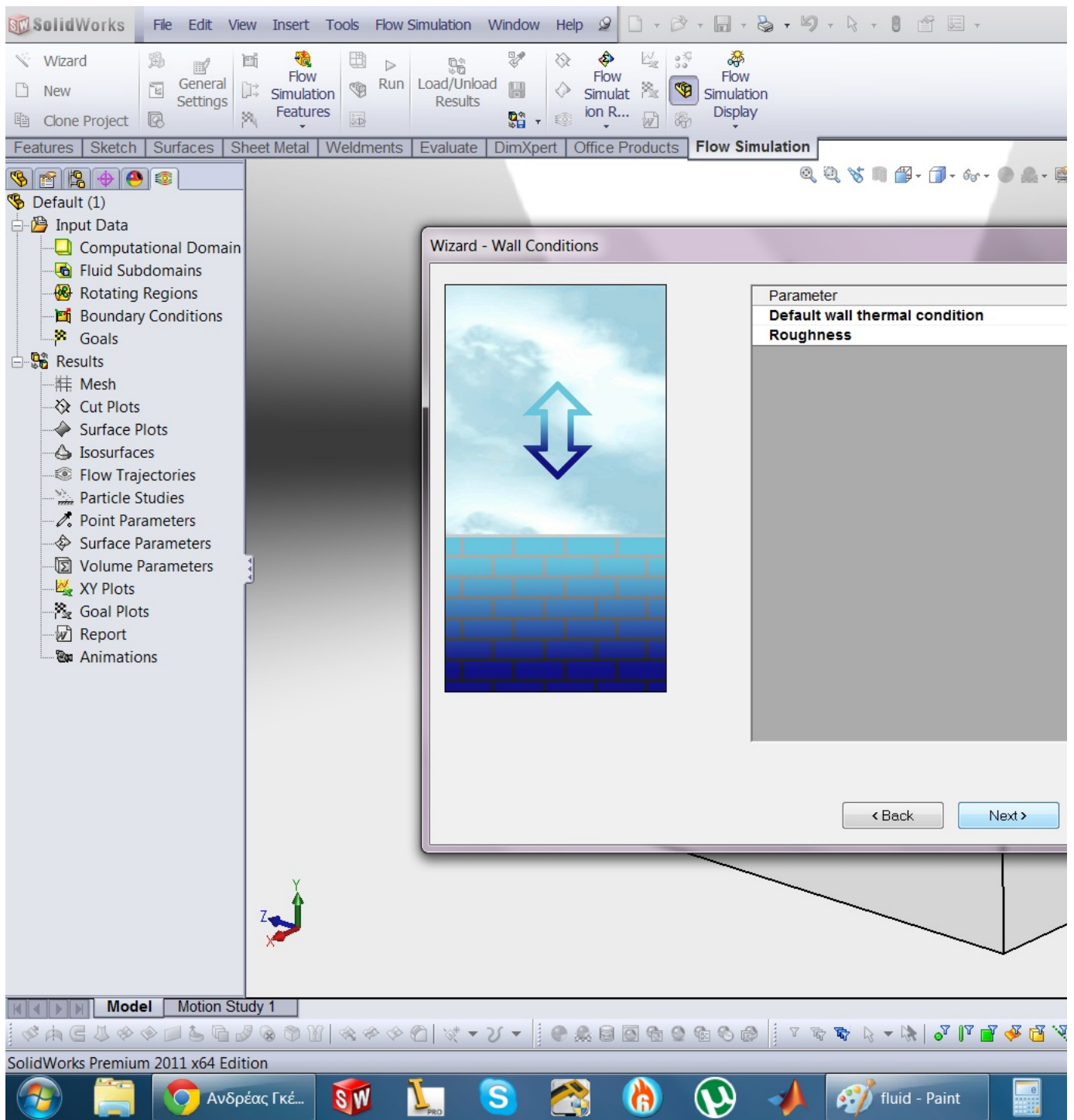
10. Step 10:

Our fluid is going to be water...



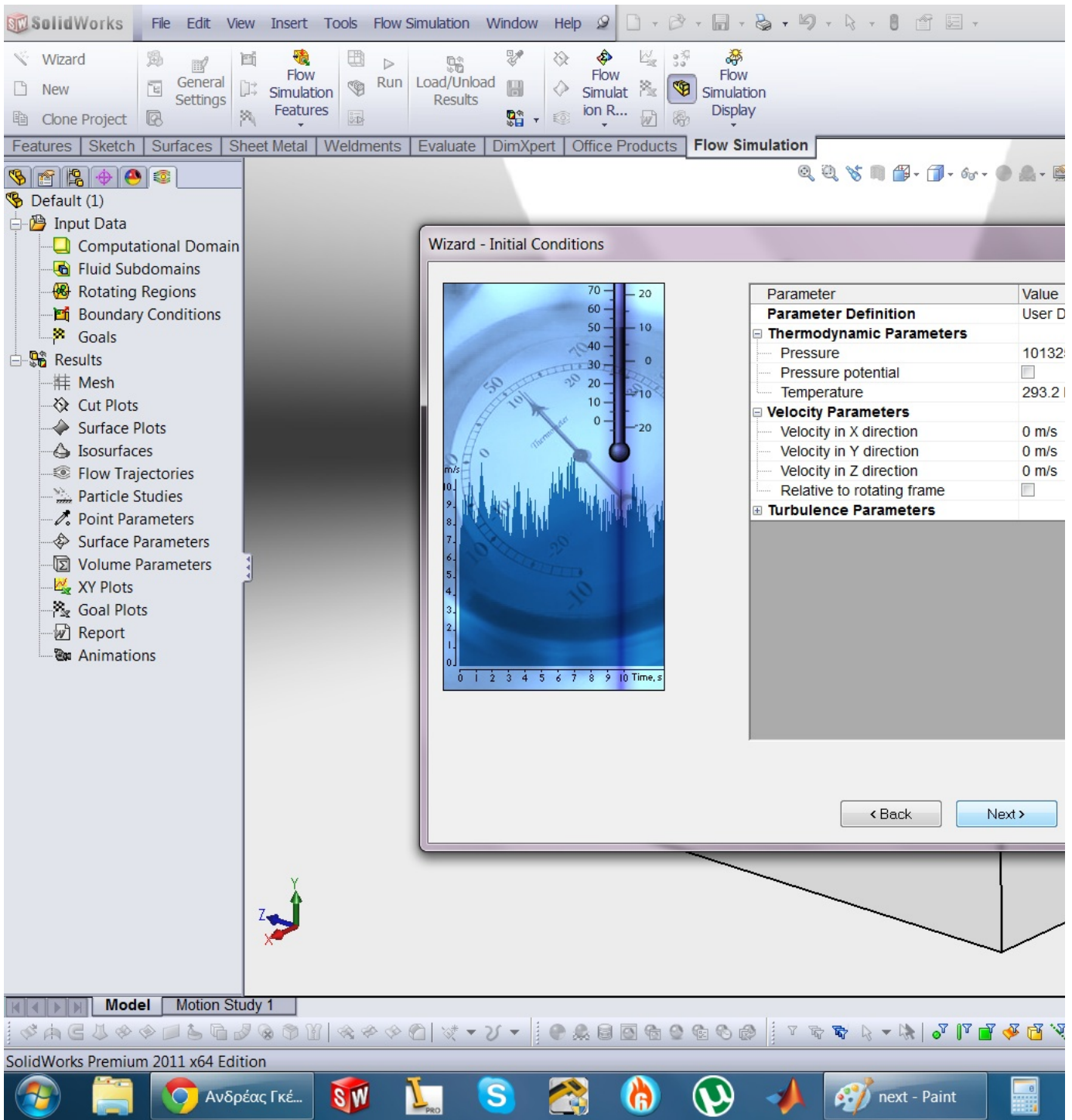
11. Step 11:

next..



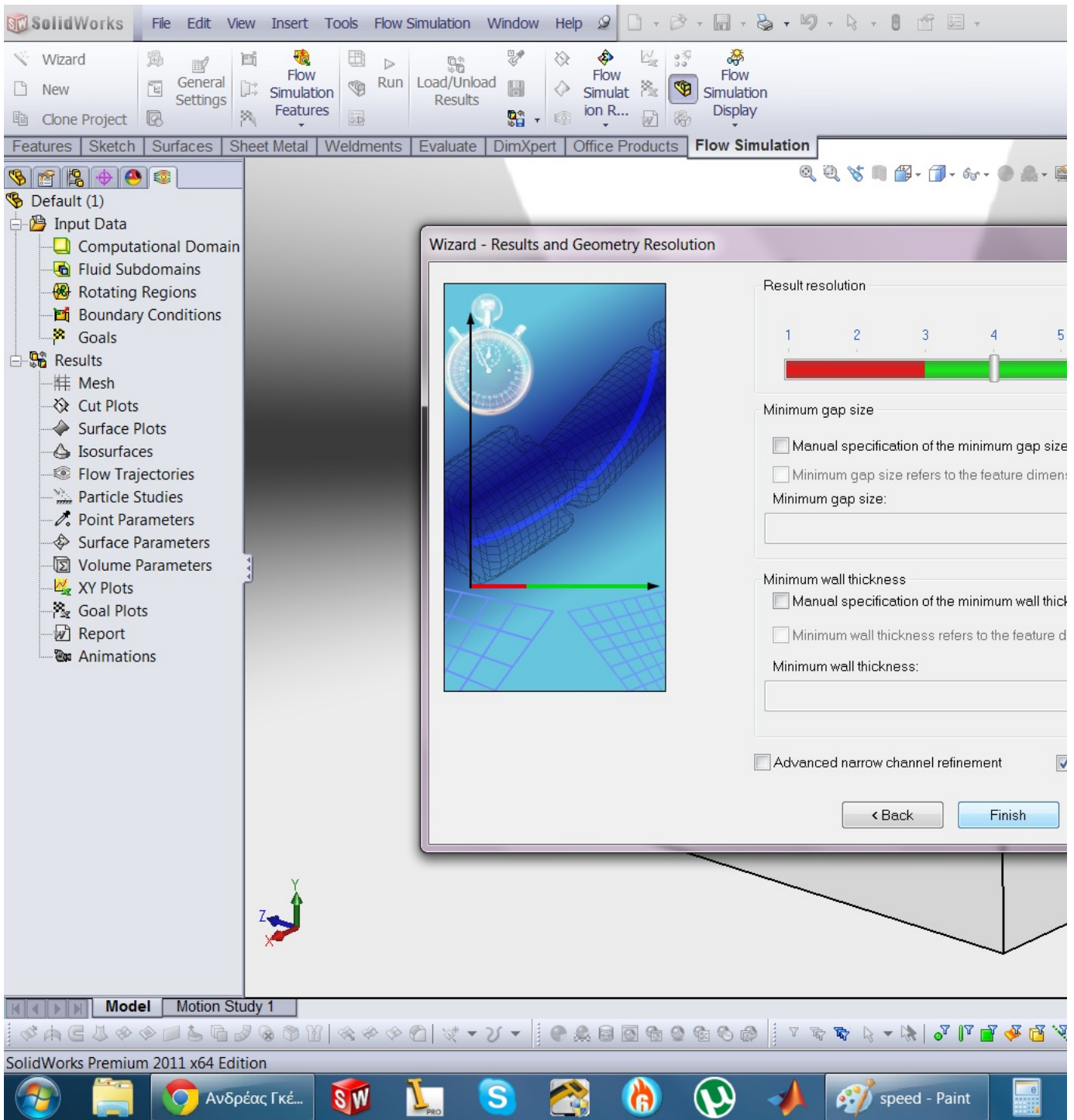
12. Step 12:

next...



13. Step 13:

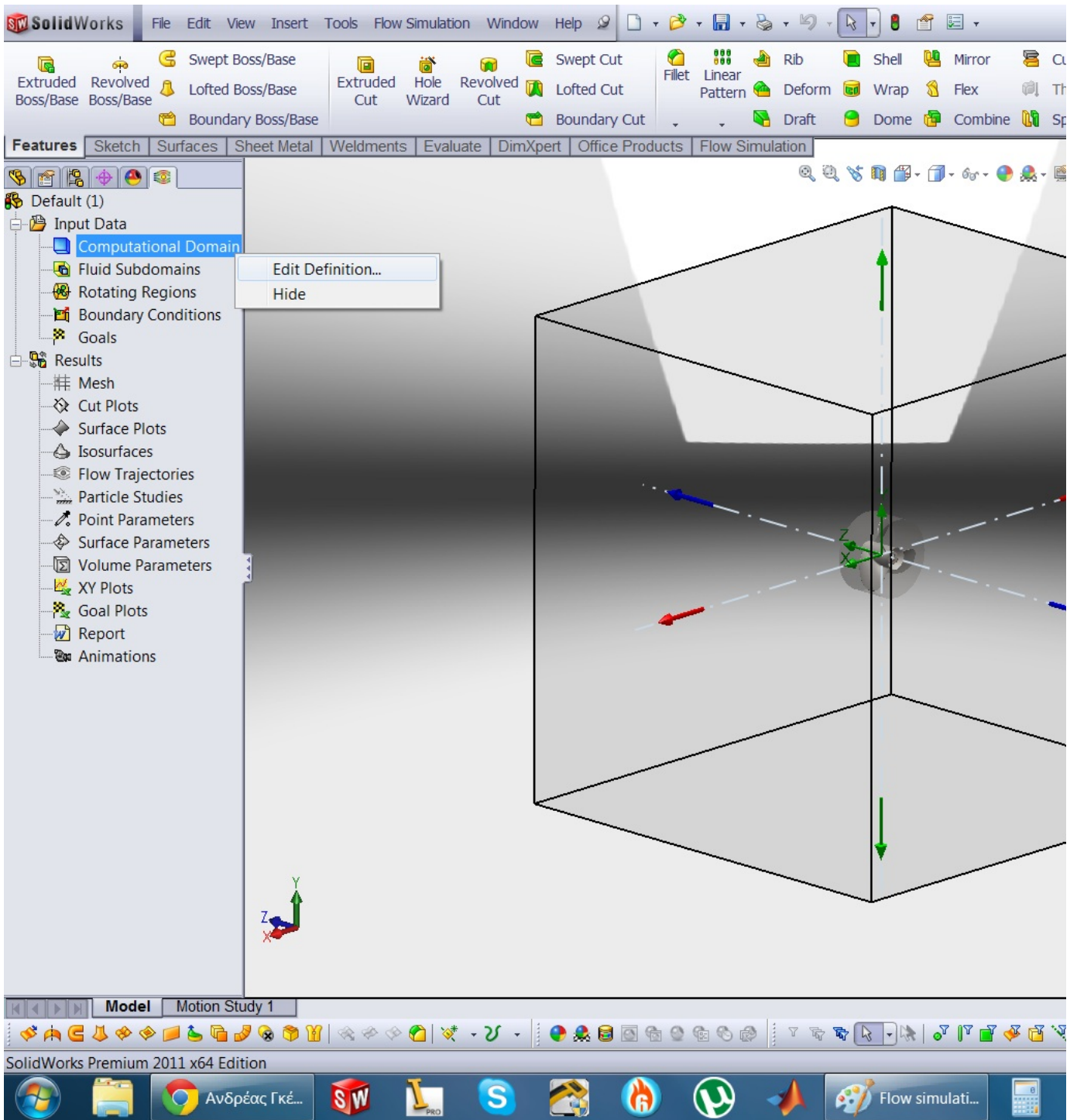
Finish... (if you want, you can adjust the resolution number on the top depending on the cpu you have)



14. Step 14:

Next we edit the computational domain... (This depends on the cpu you have... if you have a powerfull pc, you can leave it at max.... but if you don;t want to wait for a lifetime, you can reduce it...)

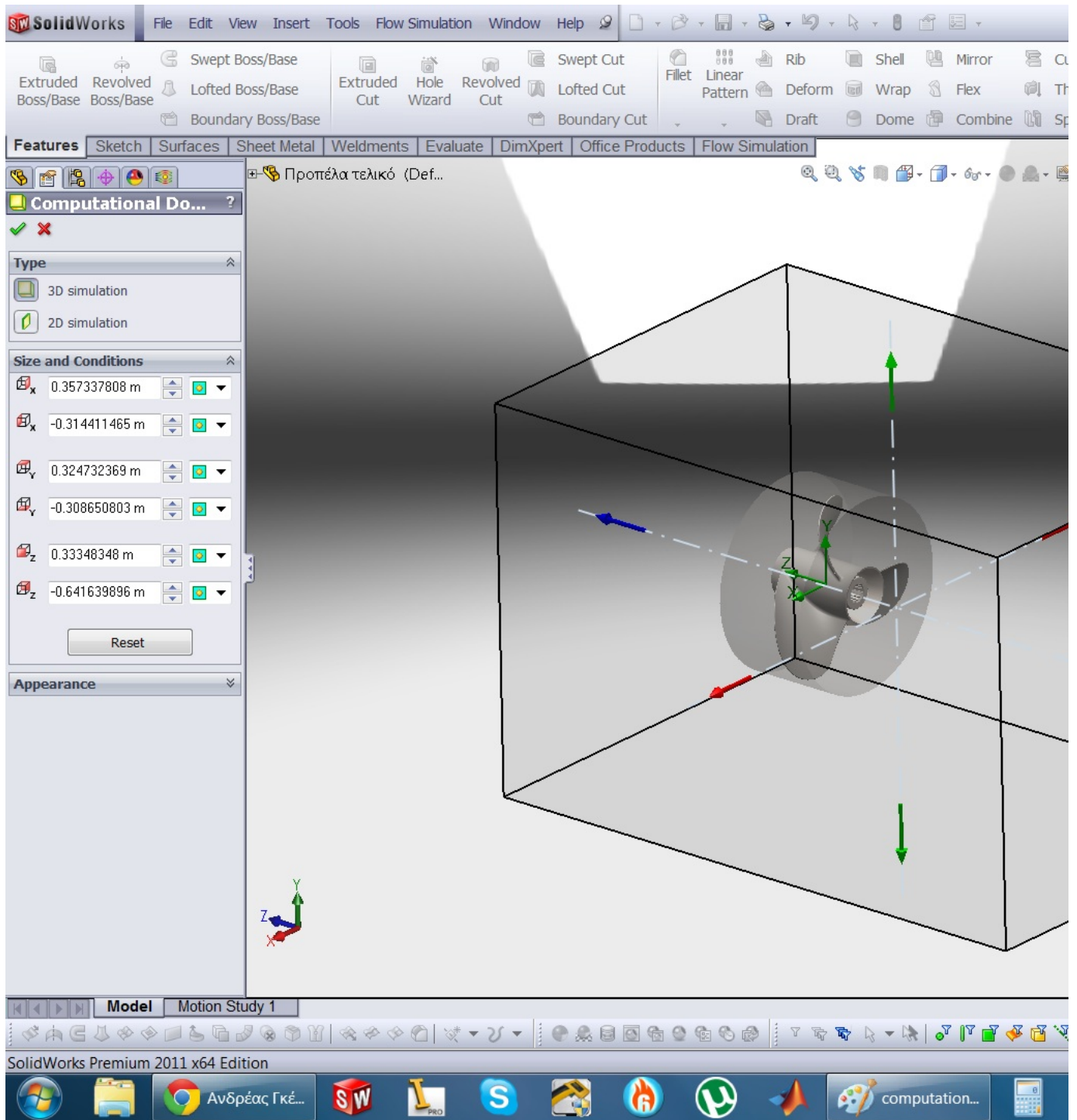




15. Step 15:

this is how i adjusted it....

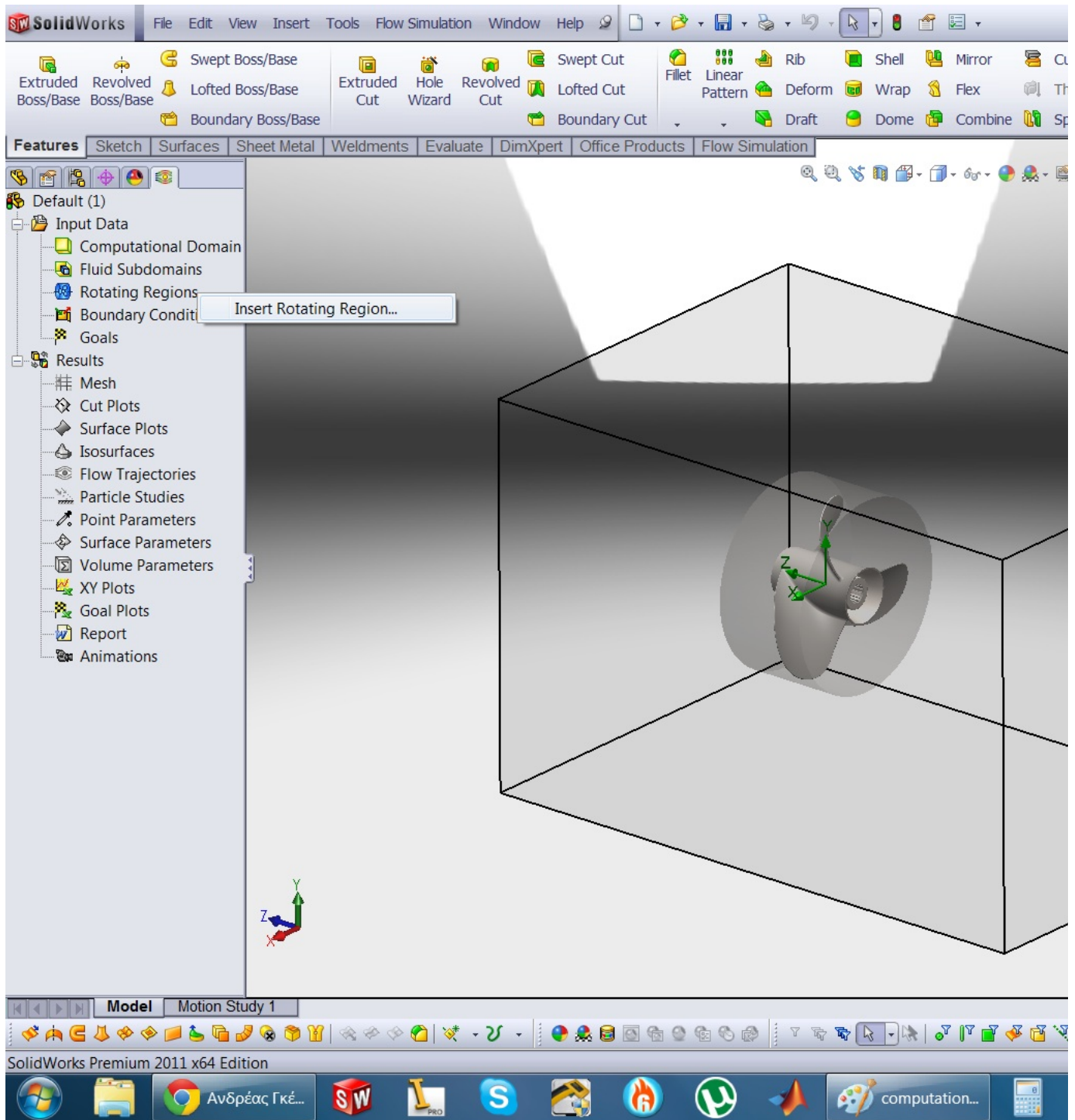




16. Step 16:

then we choose rotating regions...

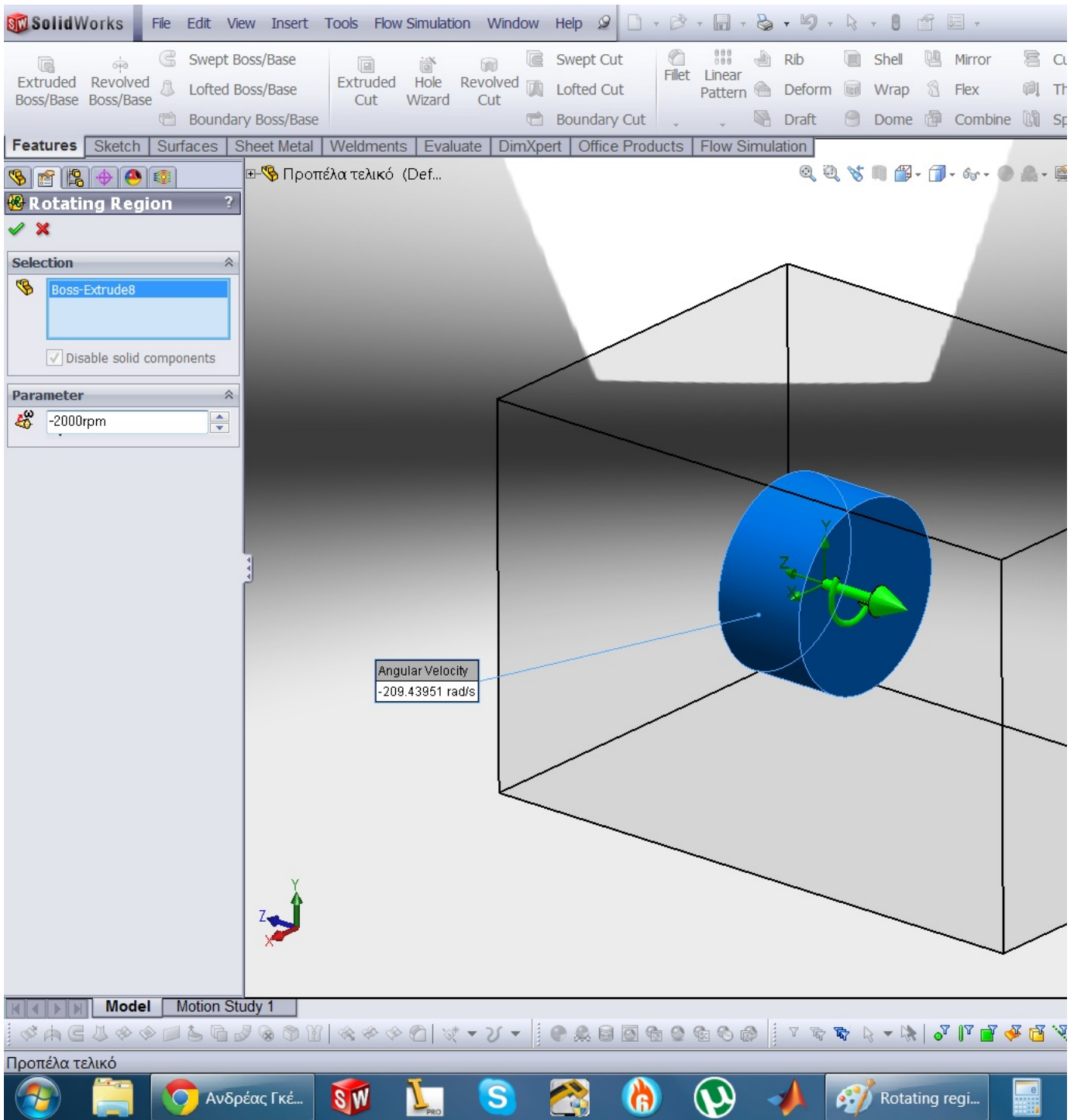




17. Step 17:

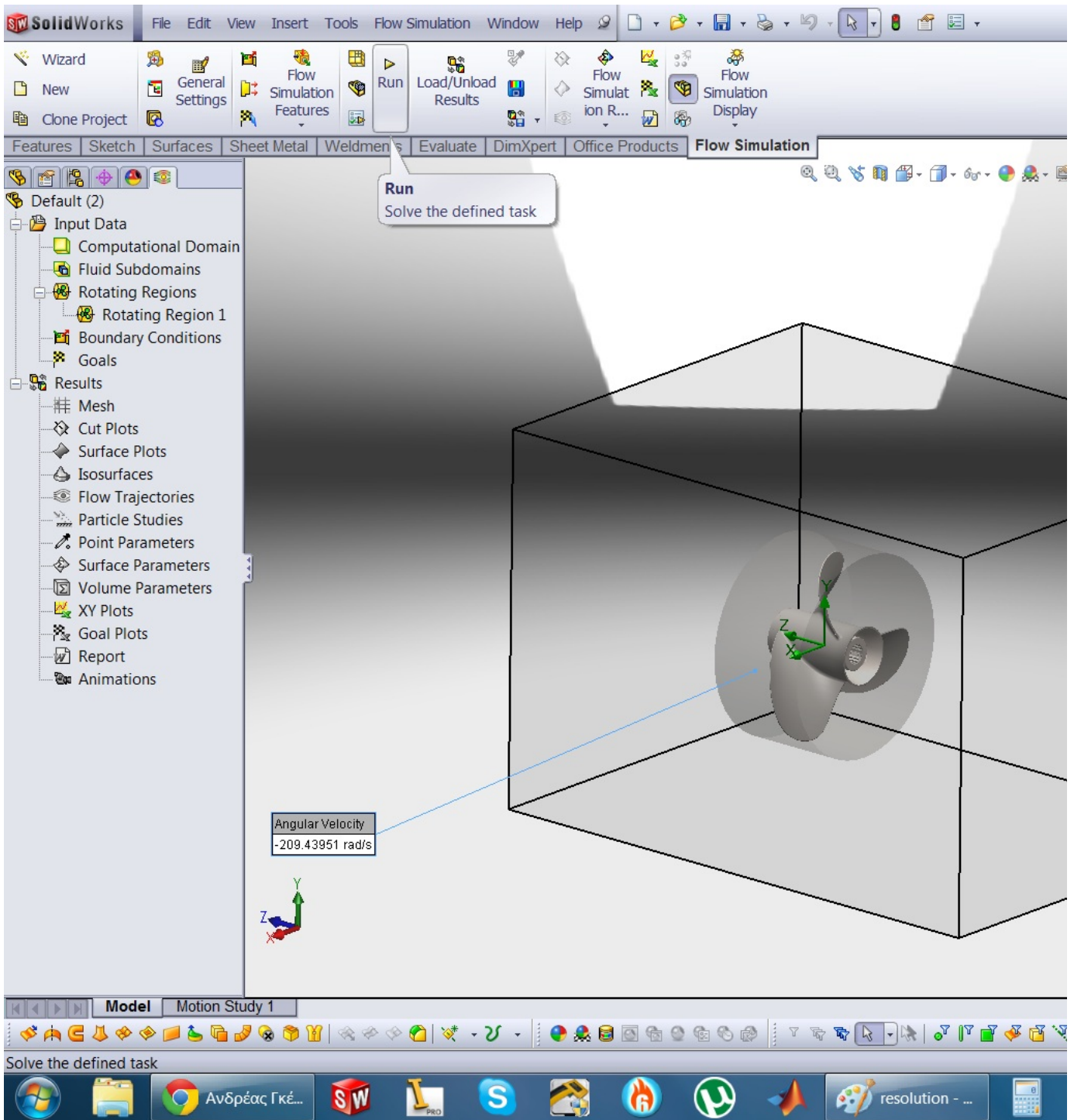
we click on the extrude body.. NOT the propeller..



**18. Step 18:**

RUN..





19. Step 19:

we wait till the solver is finished...



The screenshot shows the SolidWorks Flow Simulation environment. The main window displays the Solver: Default (2) (Προπέλα τελικό.SLDPRT) window, which is open over the main model view. The window contains an 'Info' tab with a table of simulation results and a 'Warning' section.

Parameter	Value
Status	Solver is finished.
Fluid cells	6441
Partial cells	865
Iterations	86
Last iteration finished	21:17:08
CPU time per last iteration	00:00:02
Travels	2.15
Iterations per 1 travel	41
Cpu time	0 : 0 : 30
Calculation time left	0 : 0 : 0

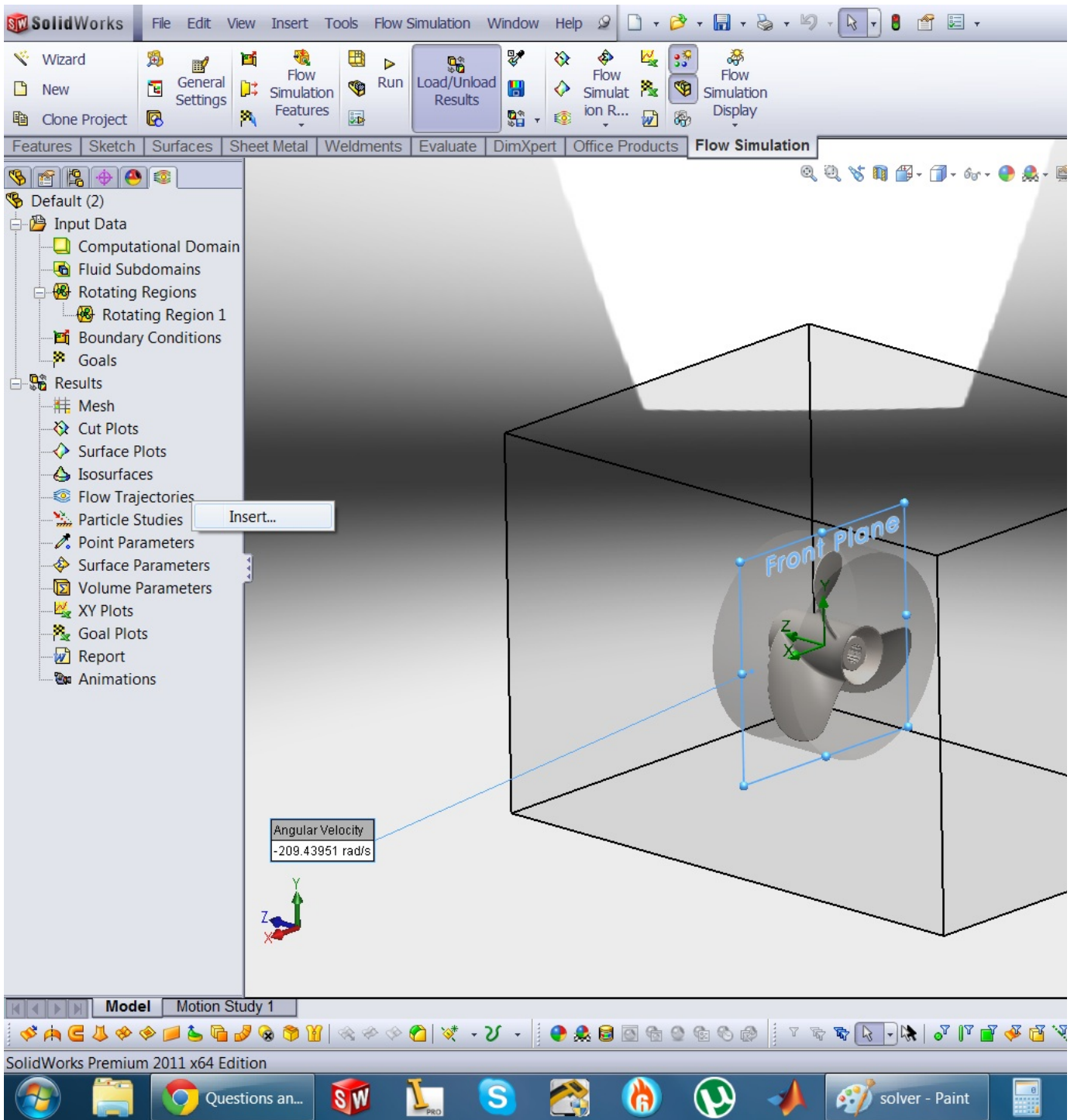
Below the 'Info' tab, there is a 'Warning' section with the following content:

Warning	Comment
No warnings	

The status bar at the bottom of the window shows 'Ready'.

20. Step 20:

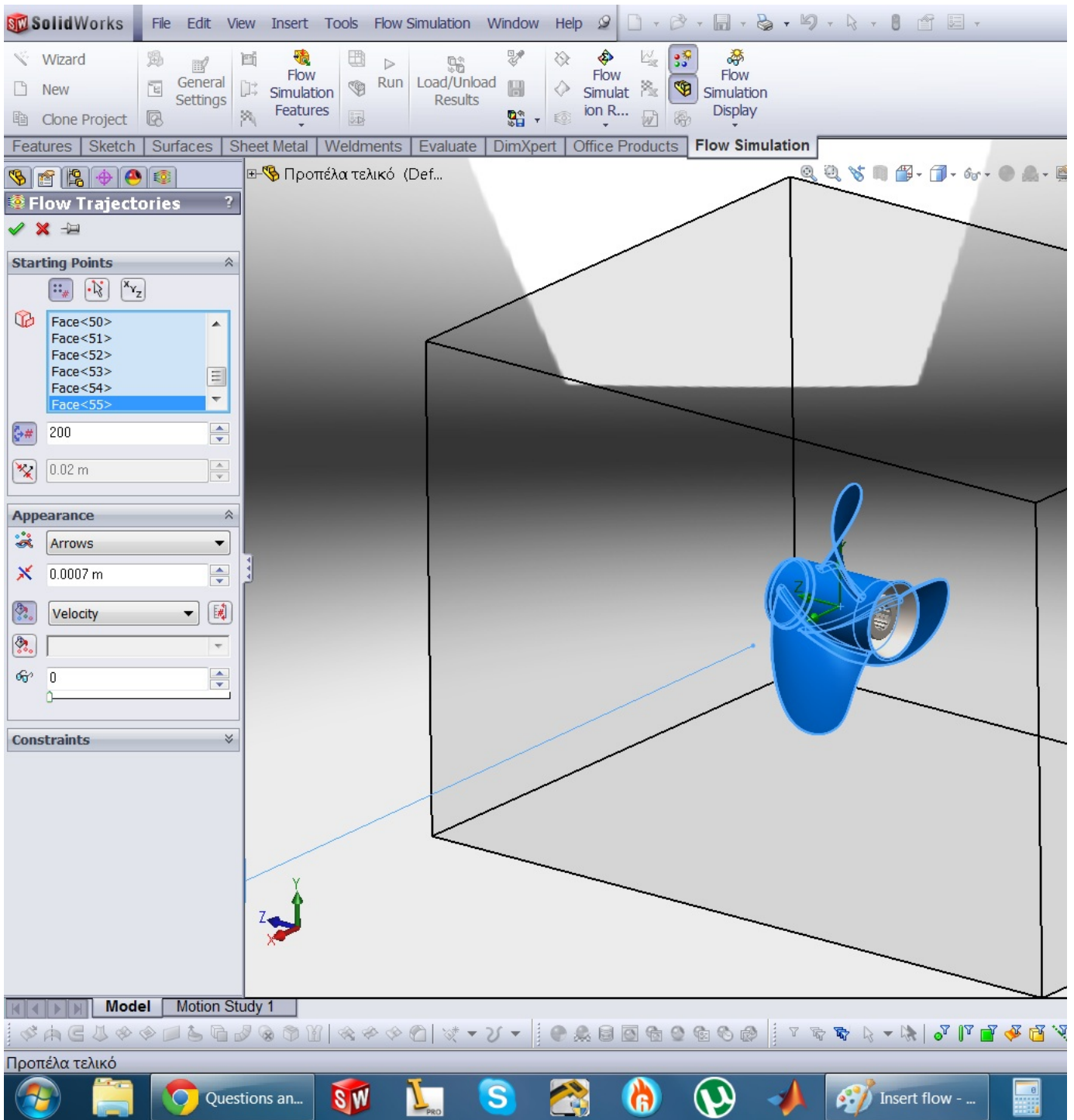
Then we go to flow trajectories...right click, insert



21. Step 21:

We select the surfaces that are in contact with the fluid and we set the number of the particles used... (Again this is up to you cpu speed.. My i7 tuned to 2.9ghz took about 4-5 minutes to do a 200 particles calculation)

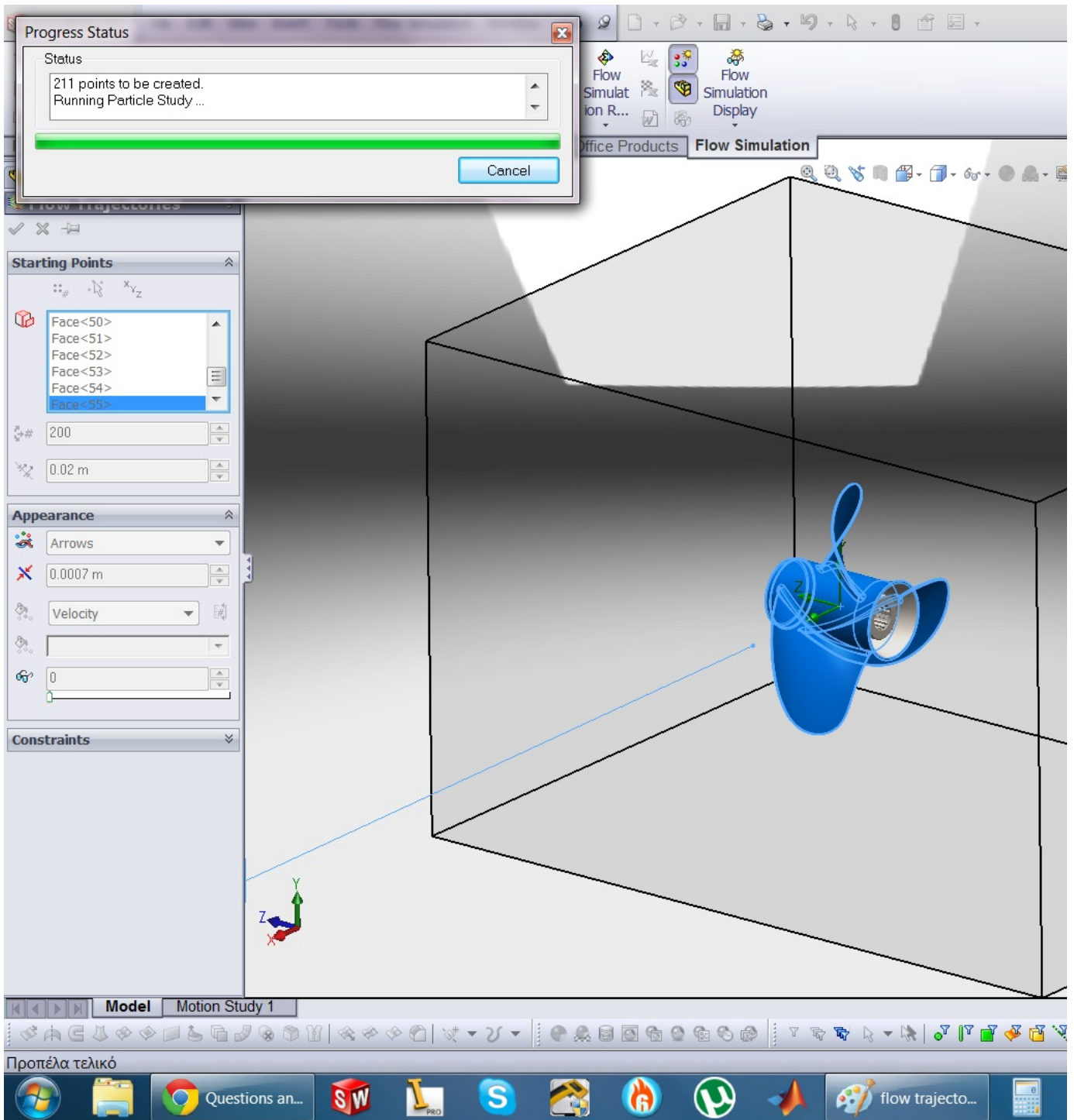




22. Step 22:

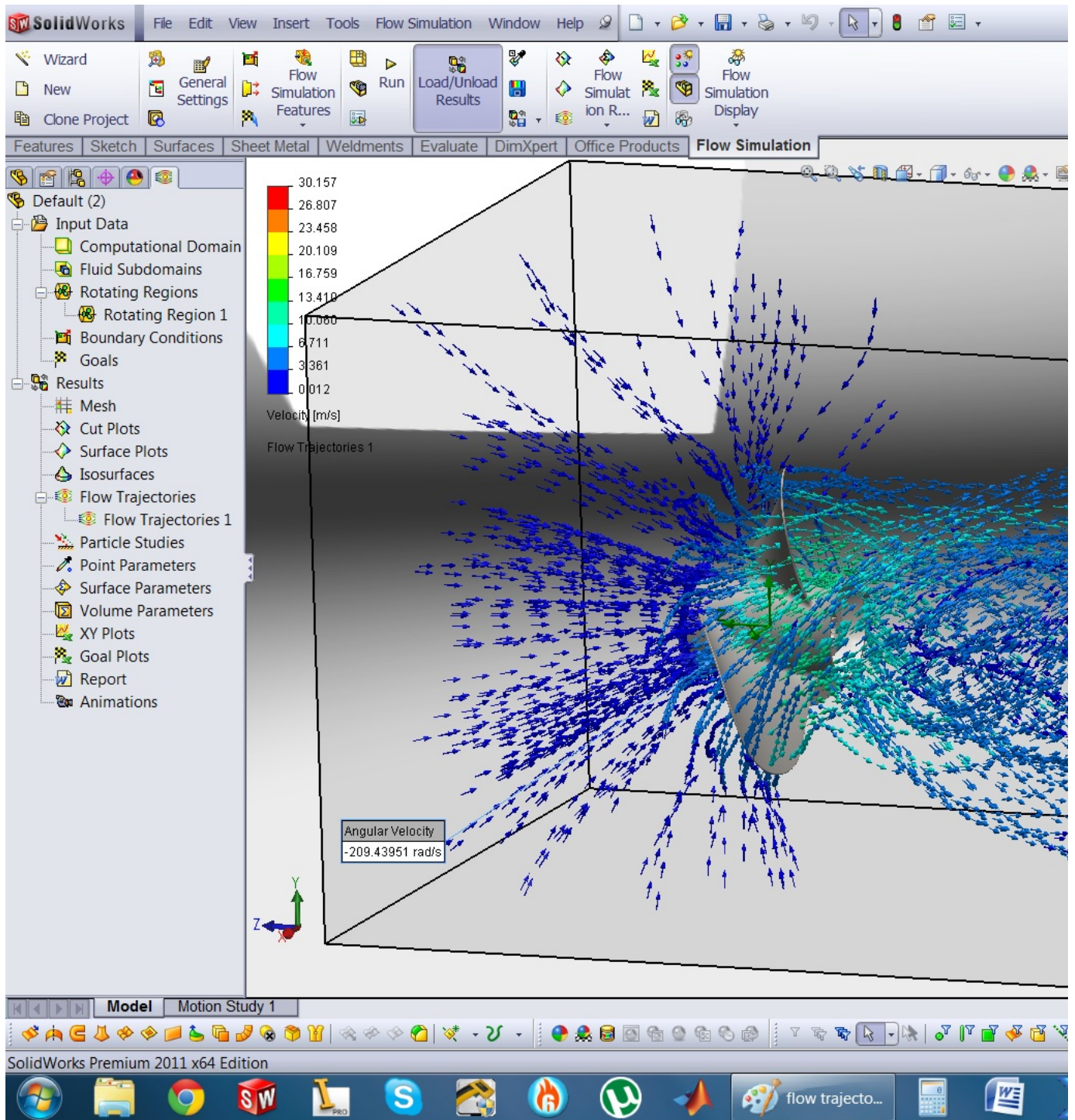
Wait till the solver is finished...





23. Step 23:

done....:)



Was this tutorial useful?

Like

Details

Skill level: Beginner

Steps: 23

Created: January 19th, 2012

Software: [SOLIDWORKS](#)

Category: [Design & CAD](#)

Tags: [solidworks](#) [water propeller flow fluid simulation tricks tips trajectories](#)

Like

Share

Share this awesome tutorial with your friends

Social

Copy link

<https://grabcad.com/tutorials/>

2432 Views

9 Likes

3 Comments

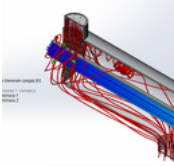
9 likes



Similar tutorials



•



•



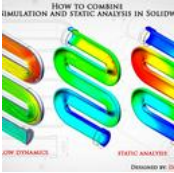
•



•



•



•

[Report this tutorial](#)

Comments

Please [log in](#) to add comments



•

[Naveen](#)

<https://www.youtube.com/watch?v=Bp-uDEjYSes&t=218s>

July 13th, 2017 17:20



•

[saumil patel](#)

it says no volume, meanse

April 9th, 2017 16:04



•

[Louis Handfield](#)

Well explained, thanks

January 30th, 2017 11:45



×

Please sign in or create an account to continue

[Sign in to GrabCAD](#) [Create an account](#)

- Community
- [Library](#)
- [Challenges](#)
- [Groups](#)
- [Questions](#)
- [Tutorials](#)
- [Engineers](#)

- Workbench
- [Overview](#)
- [Features](#)
- [Compare](#)

- Print
- [Overview](#)
- [Features](#)

- Resources
- [Blog](#)
- [Resource Center](#)
- [Help Center](#)

- About us
- [Company](#)
- [Jobs](#)

info@grabcad.com [Website](#) [Terms of Use](#) [Software Terms of Use](#) [Privacy policy](#) [Twitter](#) [Instagram](#) [Facebook](#) [LinkedIn](#)

© 2018 GrabCAD, a STRATASYS solution

The Computer-Aided Design ("CAD") files and all associated content posted to this website are created, uploaded, managed and owned by third party users. Each CAD and any associated text, image or data is in no way sponsored by or affiliated with any company, organization or real-world item, product, or good it may purport to portray.

