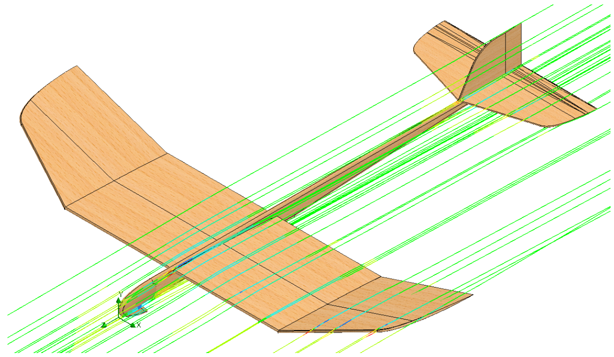
Chapter 11

Glider

Flow Sim



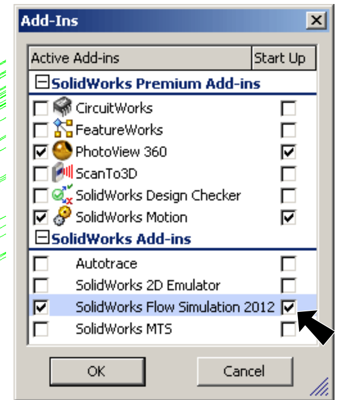


Fig. 1

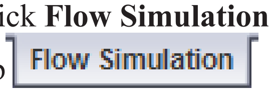
A. Add-In.

Step 1. If necessary, open your **ASSEMBLY** file.

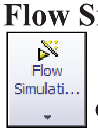
Step 2. Click Tools Menu > Add-Ins.

Step 3. In the dialog box, scroll down to **Flow Simulation** and place a check in the check box under **Active Add-Ins** and **Start-Up**, **Fig. 1**. Click OK.

B. Create Project.

Step 1. Click **Flow Simulation** tab 

on the Command Manager toolbar.

Step 2. Click **Flow Simulation Main** 

on the Flow Simulation toolbar and **Wizard** 

from the menu.

Step 3. Key-in **2 Degrees** for Configuration name, **Fig. 2**.

Step 4. Click Next.

Step 5. **Unit System:** Select **SI (m-kg-s)** for Unit system, **Fig. 3**.

Step 6. **Under Parameter**, set **Velocity** units to **Mile/hour**, **Fig. 3**.

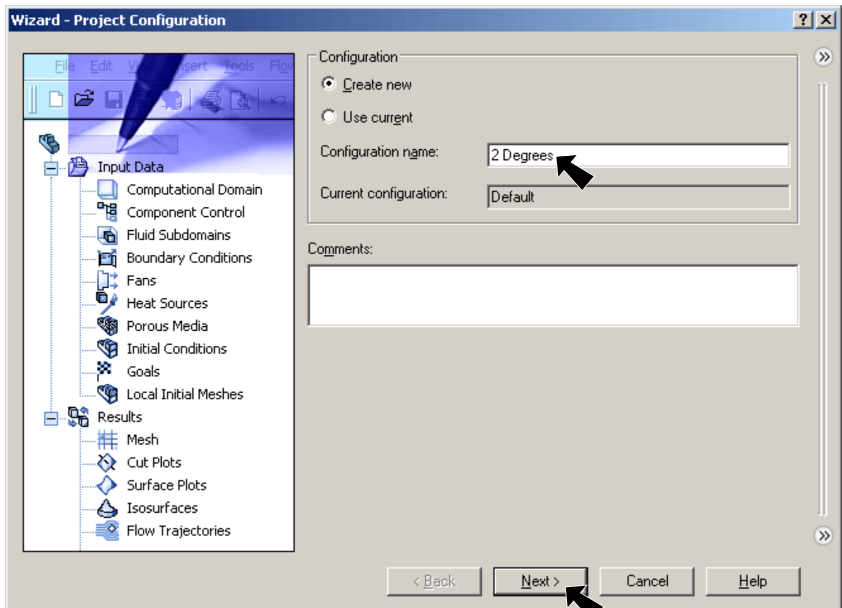


Fig. 2

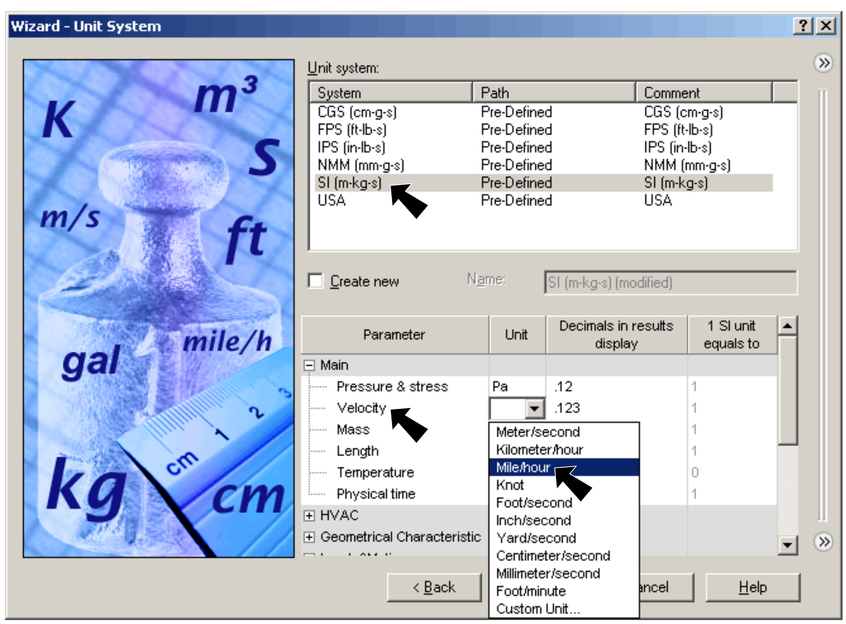


Fig. 3

Step 7. Scroll down and expand **Loads & Motion**. set **Force** units to **Gram force**, **Fig. 4**. Gram force unit is p.

Step 8. Click Next.

Step 9. **Analysis Type:** Under Analysis type, select **External**, **Fig. 5**.

Step 10. Check **Exclude cavities without flow conditions** and **Exclude internal space**, **Fig. 5**.

Step 11. Set **Reference axis to Z**, **Fig. 5**.

Step 12. Click Next.

Step 13. **Default Fluid:** Expand Gases, select **Air** and click **Add**, **Fig. 6**.

Step 14. Click Next.

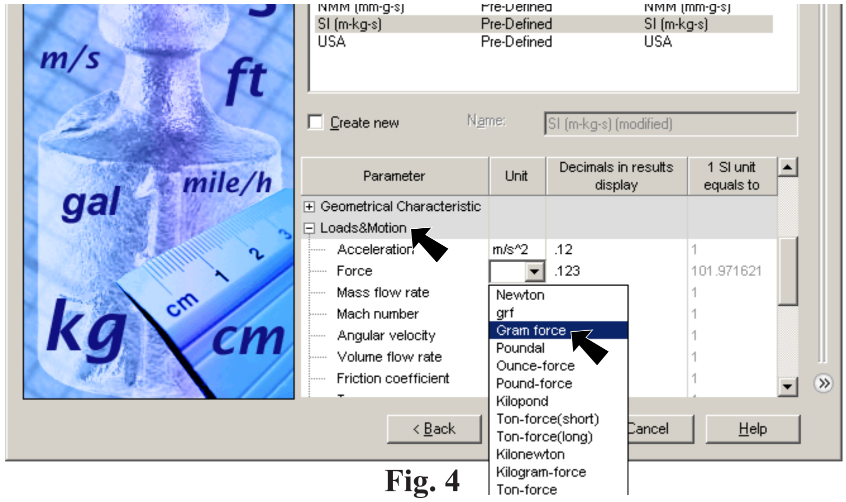


Fig. 4

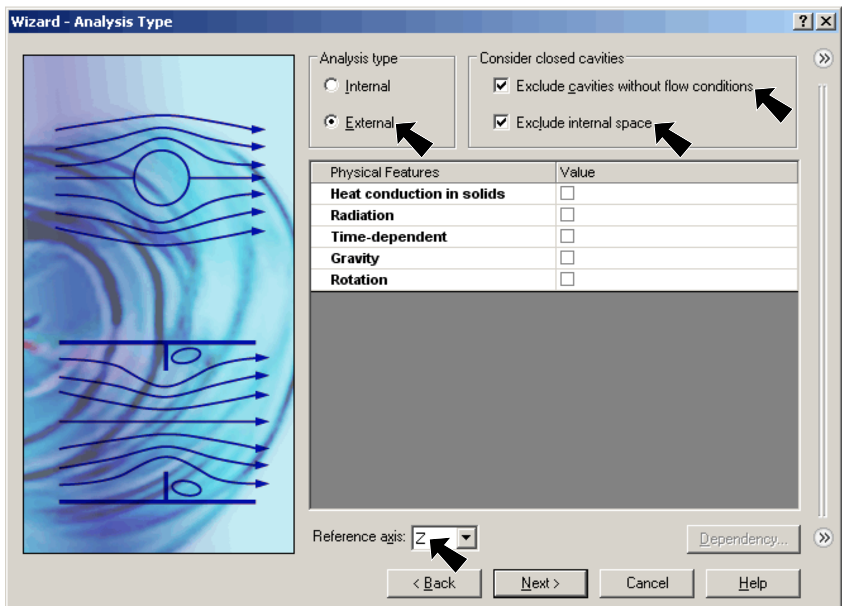


Fig. 5

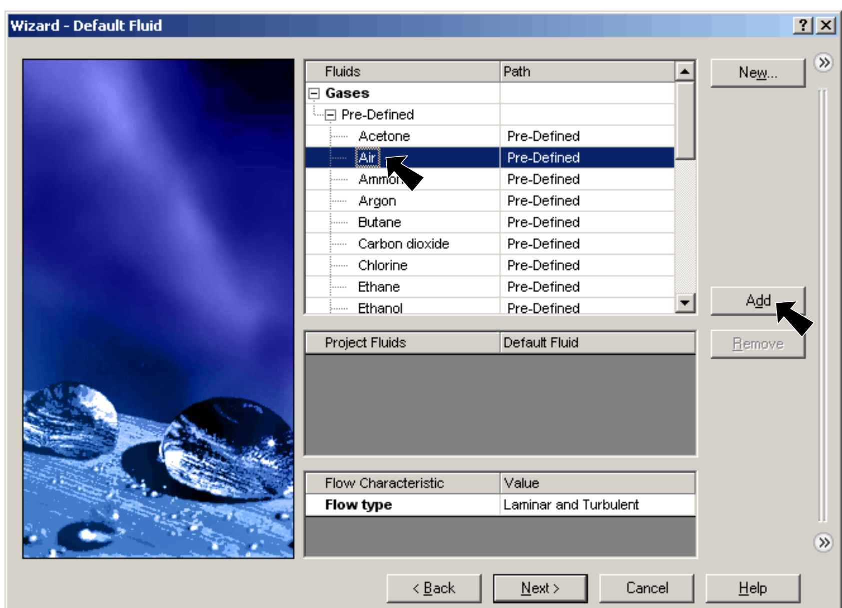


Fig. 6

Step 15. **Wall Conditions:** Use the default values for wall condition, **Fig. 7**.

Step 16. Click Next.

Step 17. **Initial Conditions:** Under Velocity parameters set **Velocity in Z direction** to **-20 mile/h**, **Fig. 8**. (click and key-in **-20**).

Step 18. Click Next.

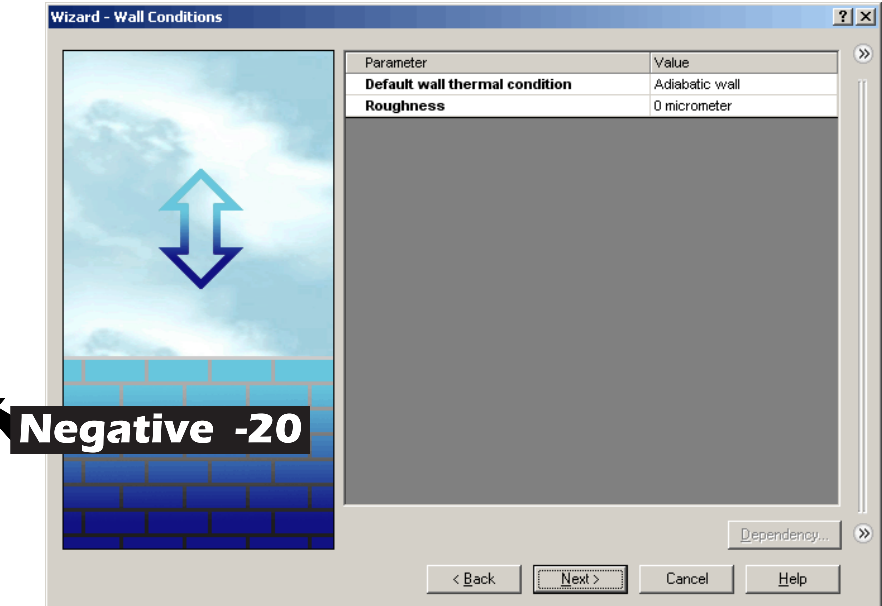


Fig. 7

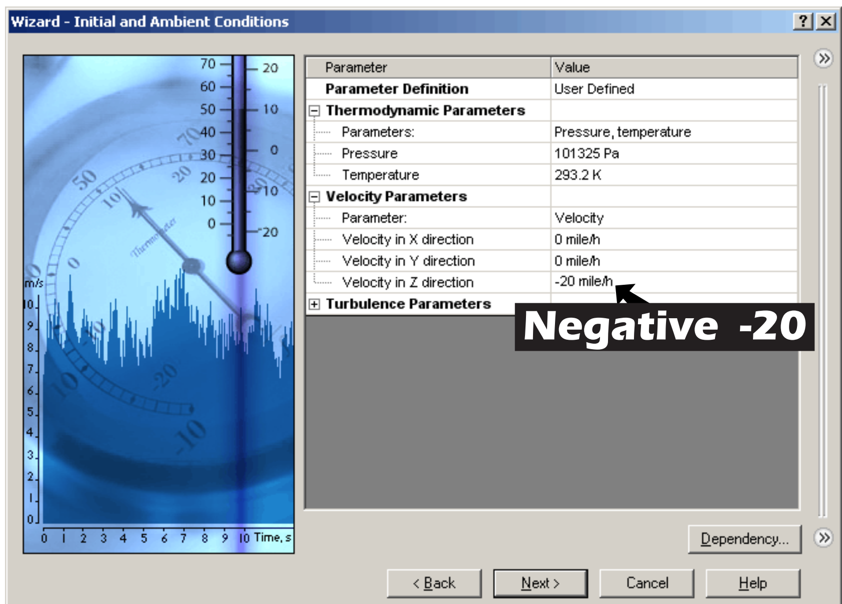


Fig. 8

Step 19. **Results and Geometry Resolution:**
Set **Result resolution to 4**, **Fig. 9**.

Step 20. Click Finish.

C. Computational Domain.

Step 1. Click Flow Simulation Menu > Computational Domain.

Step 2. Set values as shown here and **Fig. 10**.

X max .23
X min 0

Y max .2
Y min -.15

Z max .3
Z min -.4

set **X min to Symmetry**, **Fig. 11**

click OK.

Step 3. Click **Front** on the Views toolbar (**Ctrl-1**). Use **Z** key to zoom out to see domain, **Fig. 13**.

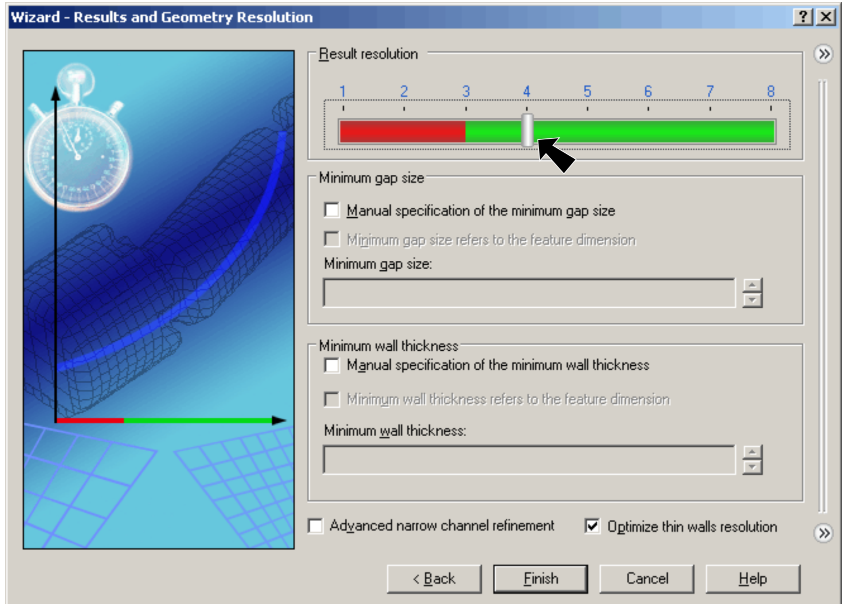


Fig. 9

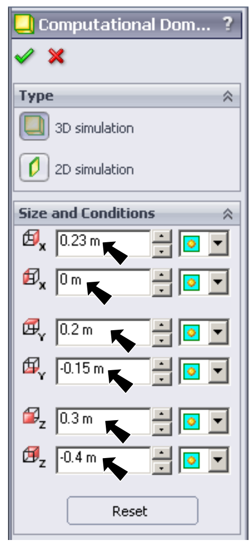


Fig. 10

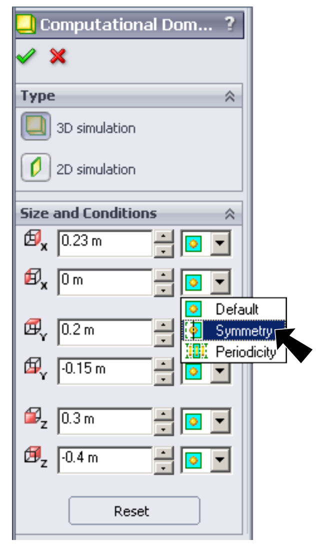


Fig. 11

Step 4. **Only half the glider** should be inside the domain as shown in **Fig. 13**. This will reduce the solver time. Solver can take several minutes.

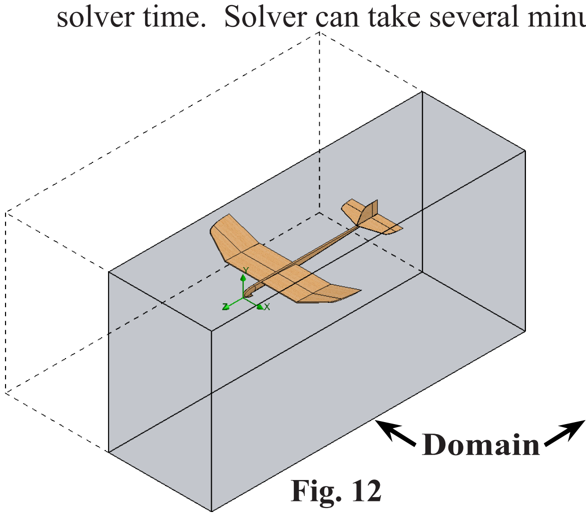


Fig. 12

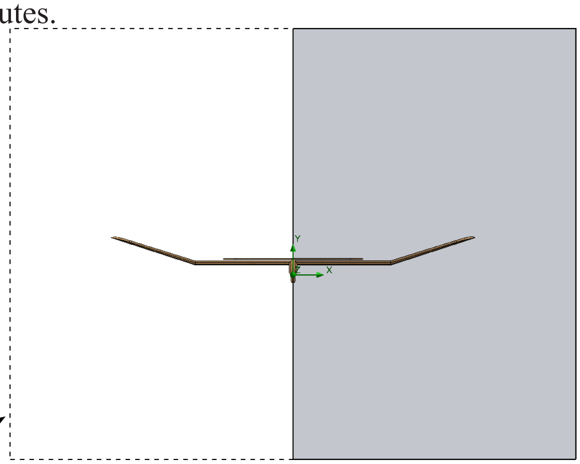


Fig. 13

D. Insert Global Goals.

Step 1. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**)

Step 2. Click Flow Simulation Menu > Insert > **Global Goals**.

Step 3. Click the **Flow Simulation tab** in the Feature Manager, **Fig. 14**.

Step 4. Drag the edge of the Feature Manager to the right to expand, **Fig. 14**.

Step 5. Scroll down the Parameters to **Force** and **check:**

Force (Y)
Force (Z) **Fig. 14**

click OK .

Step 6. Rename the goals to **Lift** and **Drag**. To rename, click and hold over **Force (Y)** and key-in **Lift**, **Fig. 15**. Rename **Force (Z)** to **Drag**.

Step 7. Save. Use **Ctrl-S**.

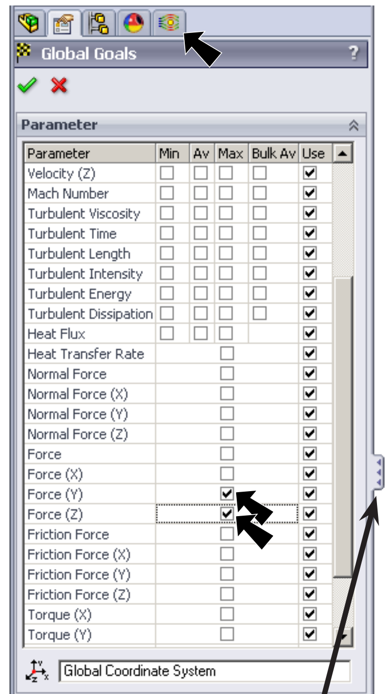


Fig. 14
Drag right

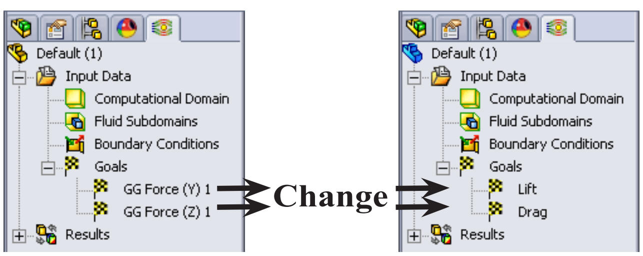


Fig. 15

E. Run Analysis.

Step 1. Click **Flow Simulation Main** on the Flow Simulation toolbar and **Run** from the menu.

Step 2. Click **Run** in the Run dialog box, **Fig. 16**.

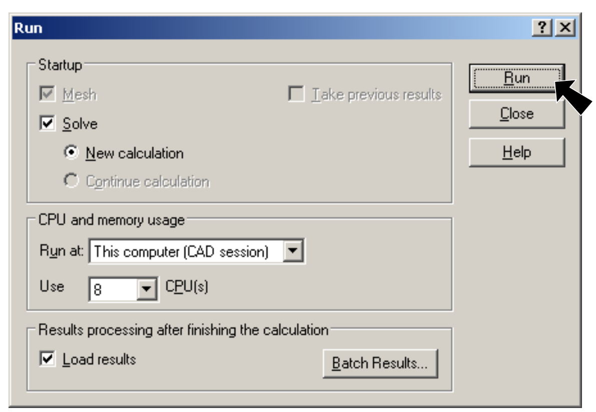


Fig. 16

F. Solver.

Step 1. In the Solver dialog box you can view Calculation time left, **Fig. 17**. The CPU time runs around 1-4 minutes depending on glider design and computer.

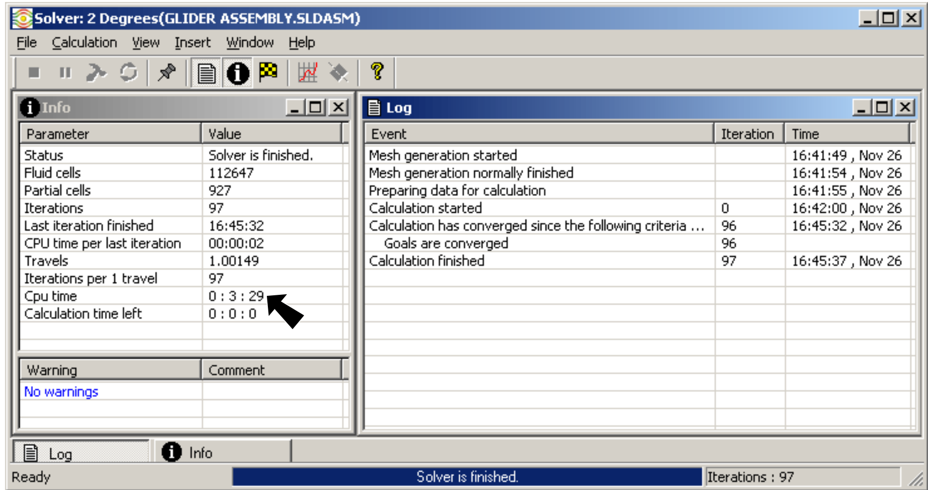


Fig. 17

Step 2. Click Insert Menu > Goals Plot in the Solver, **Fig. 18**.

Step 3. Click **Add All** and OK in Add/Remove Goals, **Fig. 18**. Our Lift was **5.19**, **Fig. 19**.

Step 4. Close the Solver dialog box, click File Menu > Close.

Step 5. Save. Use **Ctrl-S**.

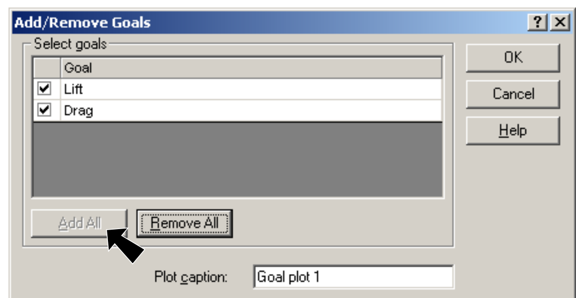


Fig. 18

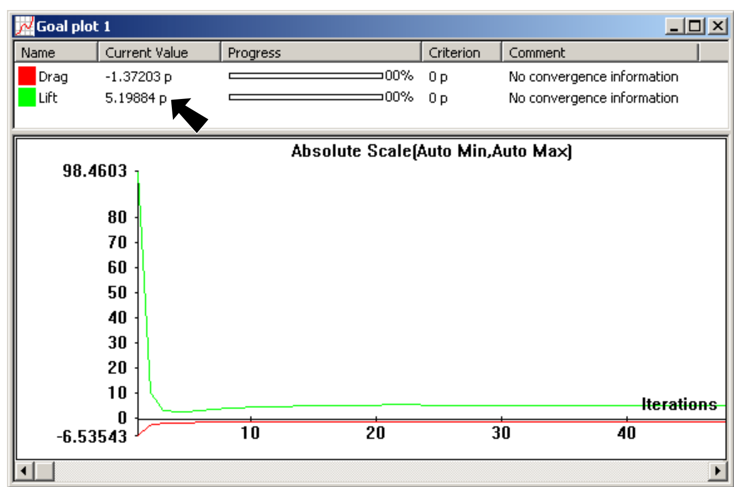


Fig. 19

G. Surface Plots.

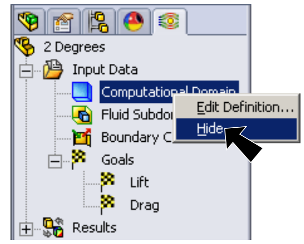
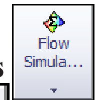
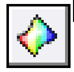


Fig. 20

Step 1. **Right** click **Computational Domain** in Flow Simulation tree and click **Hide**, **Fig. 20**.

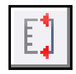
Step 2. Click **Flow Simulation Results Features** 

on the Flow Simulation toolbar and **Surface Plot**  from the menu.

Step 3. In the Property Manager set:

check **Use all faces**, **Fig. 21**

select **Contours**

click **Adjust Minimum and Maximum** 

Maximum  pressure **101365**

Minimum  pressure **101300**

click OK .

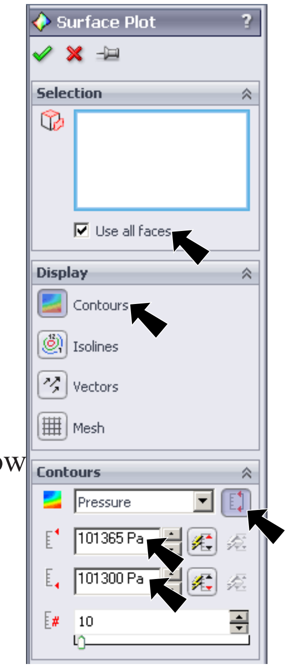


Fig. 21

Step 4. After viewing the surface plot, expand **Results** and Surface Plots in Flow Simulation tree, **right click Surface Plots 1** and click **Hide**, **Fig. 23**.

Step 5. Save. Use **Ctrl-S**.

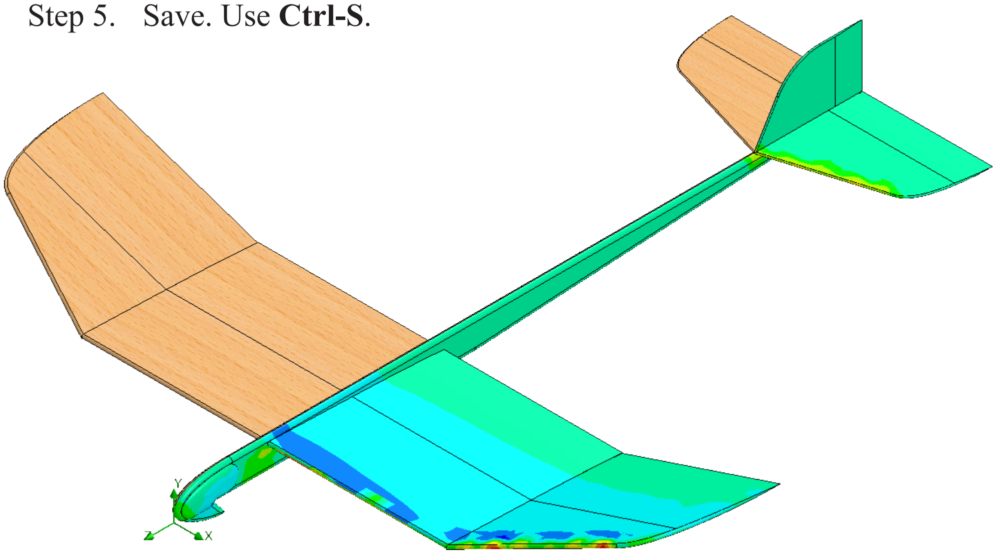


Fig. 22

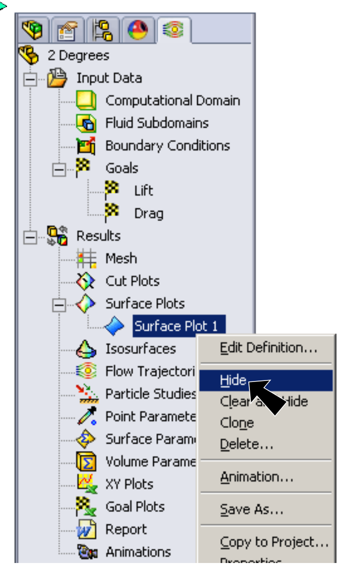


Fig. 23

H. Flow Trajectories.

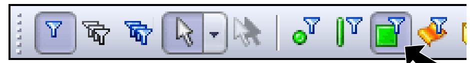


Fig. 24

Step 1. Click **Filter Faces** (**X**) on the **Selection Filter toolbar** at the bottom of the display, **Fig. 24**. If necessary, use **F5** key to display toolbar.

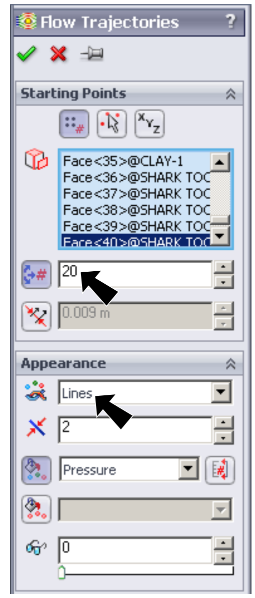


Fig. 25

Step 2. Click **Flow Simulation Results Features** (Flow Simula...) on the Flow Simulation toolbar and **Flow Trajectories** from the menu.

Step 3. In the Flow Trajectories Property Manager:

under Starting Points, **Fig. 25**

Select References. To select References**, right click a face of fuselage and click Select Tangency** from menu, **Fig 26**. Repeat, right click and select Tangency on, Wings, H Stab, V Stab, Clay and Shark Tooth, **Fig 27**.

set **Number of Trajectories** **20**, **Fig. 25**

under Options

set **Draw Trajectories As** **Lines**

click OK, **Fig. 28**.

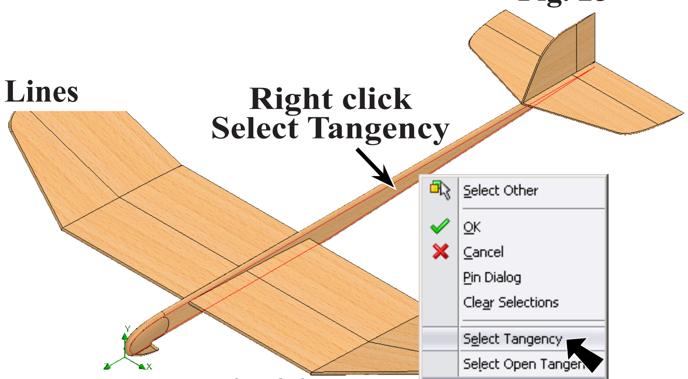


Fig. 26

Step 4. After viewing the flow trajectories, **right** click **Flow Trajectories 1** in Flow Simulation tree and click **Hide**, **Fig. 29**.

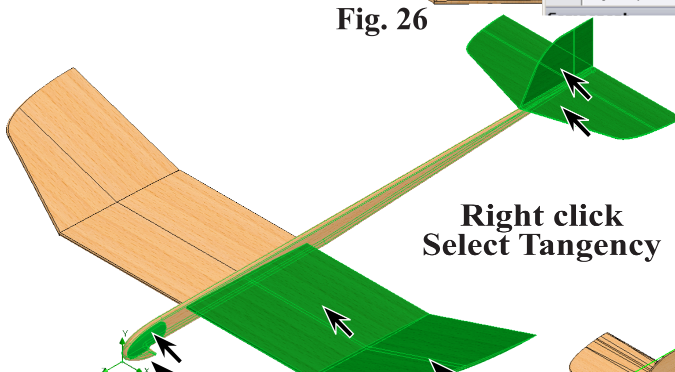


Fig. 27

Step 5. **Turn off Filter Faces** (**X**) on the **Selection Filter toolbar** at the bottom of the display, **Fig. 24**.

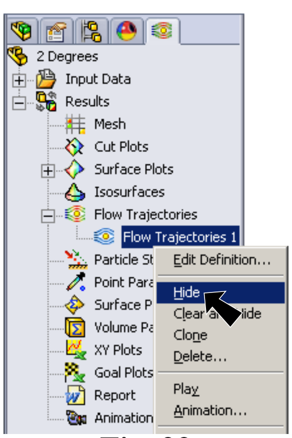


Fig. 29

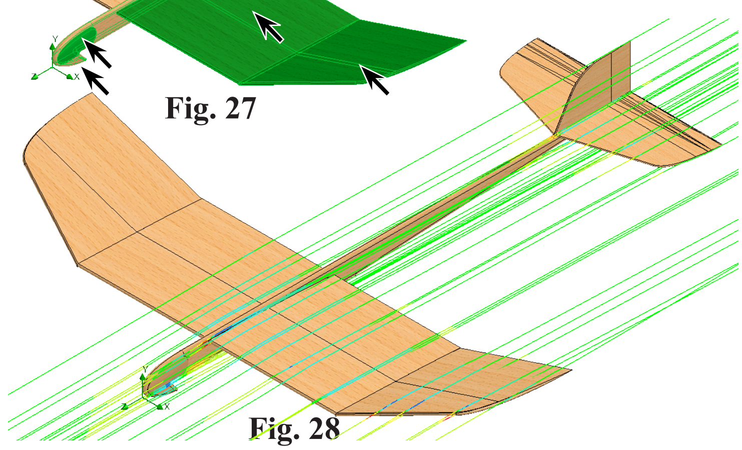


Fig. 28

I. Goal Plot.

Step 1. Click Flow Simulation Menu > Results > **Goal Plot**.

Step 2. In the Goal Plot Property Manager set:

check **All**

click OK, **Fig. 30**.

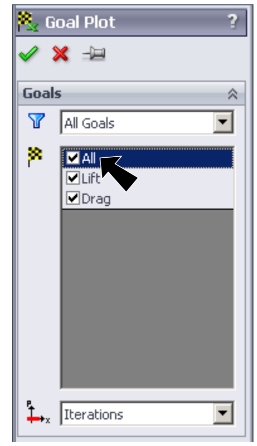


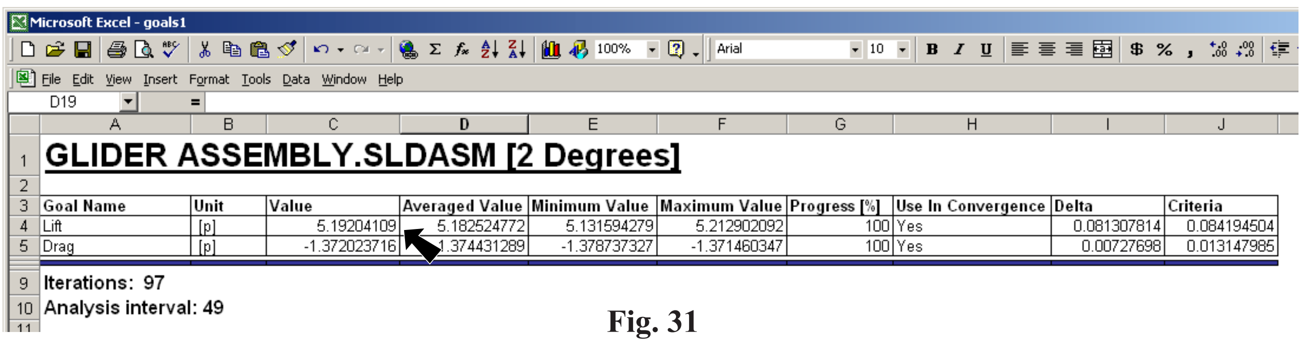
Fig. 30

Step 3. An Excel file is opened.

What is your **Lift?** Is it more than **31.24**?, **Fig. 31**.

Click the Force (Y) tab at the bottom of the Excel file to view the chart, **Fig. 32**.

Close the Excel file.



GLIDER ASSEMBLY.SLDASM [2 Degrees]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Lift	[p]	5.19204109	5.182524772	5.131594279	5.212902092	100	Yes	0.081307814	0.084194504
Drag	[p]	-1.372023716	-1.374431289	-1.378737327	-1.371460347	100	Yes	0.00727698	0.013147985

Iterations: 97

Analysis interval: 49

Fig. 31

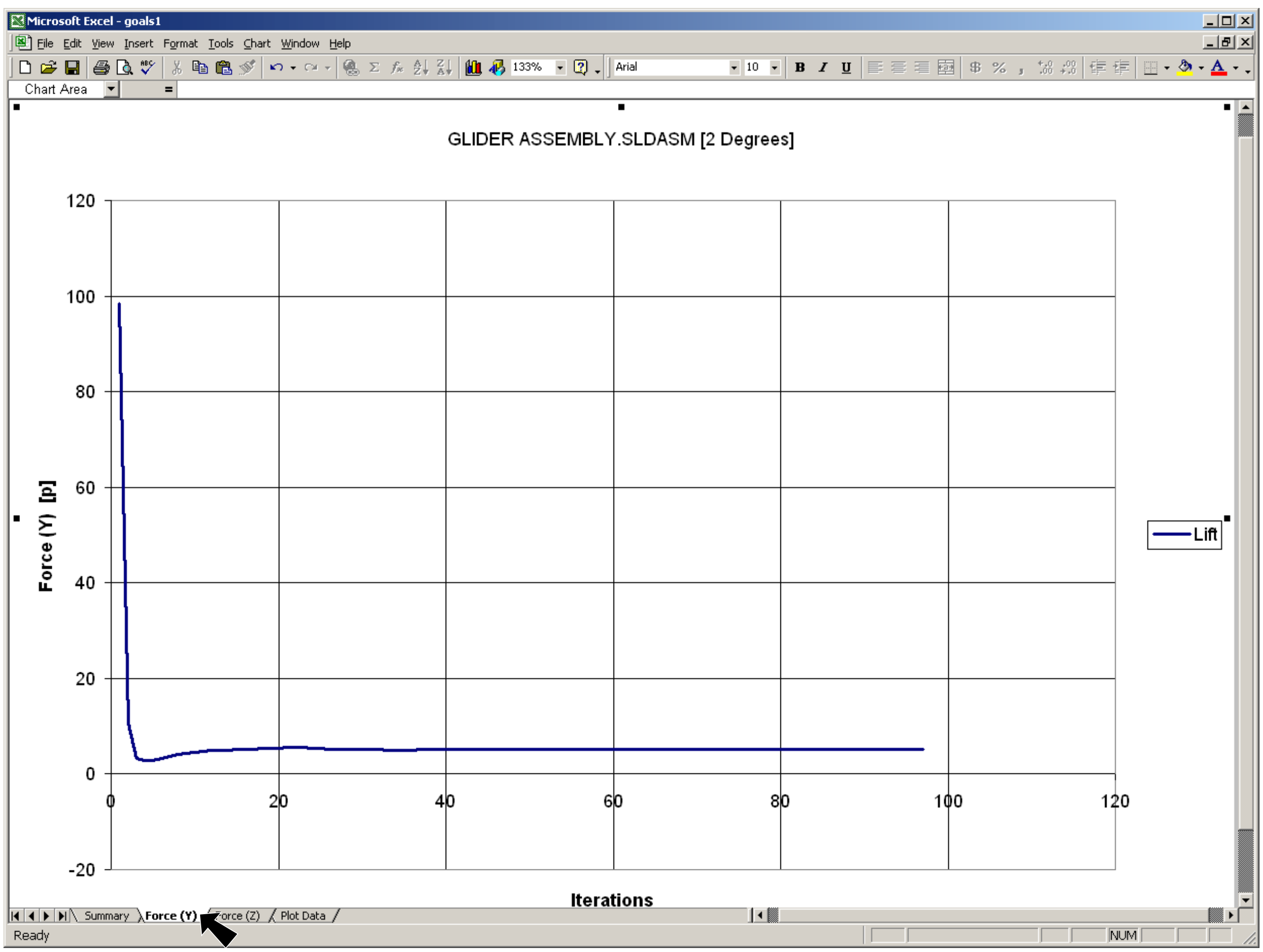


Fig. 32

J. Animate Flow Trajectories.

Step 1. Click **Right**  on the Standard Views toolbar. (**Ctrl-4**)

Step 2. **Right** click **Flow Trajectories 1** in Flow Simulation tree and click **Animation**, **Fig. 33**.

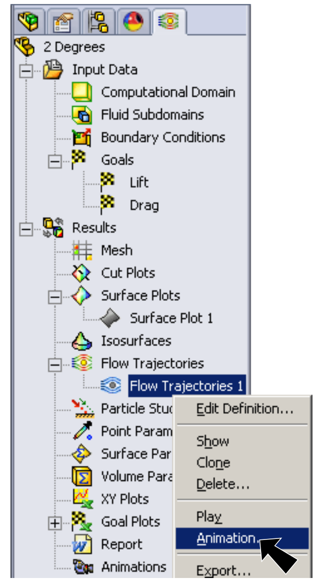


Fig. 33

Step 3. At the bottom right of the display in the animation control panel, click **Expand**  to expand the panel.

Step 4. Click **Play**  in animation control panel, **Fig. 34**.

Step 5. **Right** click **Flow Trajectories 1** in the Animation tree and click **Edit Definition**, **Fig. 34**.

Step 6. In the Flow Trajectories Property Manager:

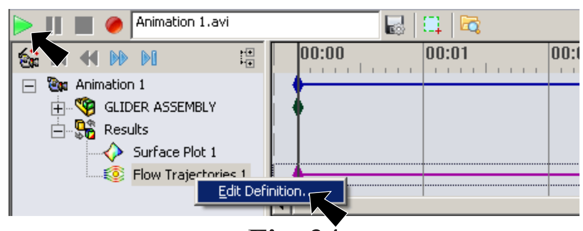
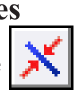


Fig. 34

under **Appearance**, **Fig. 35**

set **Draw Trajectories As**  to **Spheres**

set **Cross Size**  to **.003**

click OK .

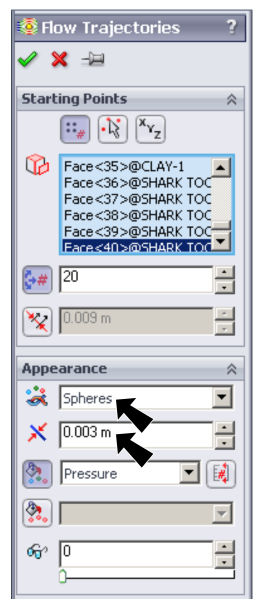


Fig. 35

Step 7. Click **Play**  in animation control panel, **Fig. 34**.

Step 8. Experiment with other Flow Trajectories setting. To change setting, **right** click **Flow Trajectories 1** and click **Edit Definition**, **Fig. 34**.

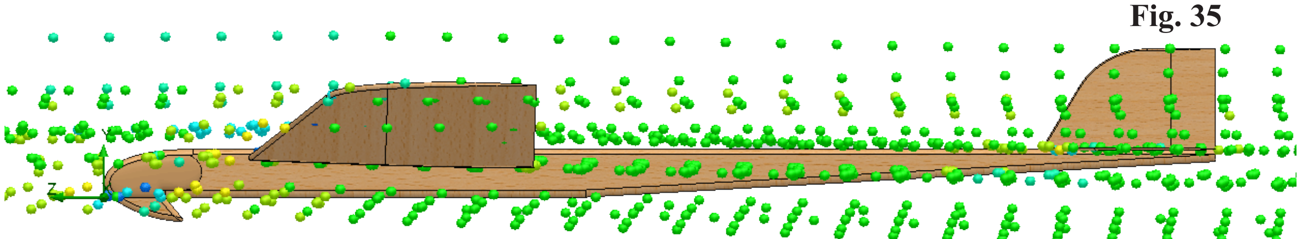


Fig. 36

K. Create Animation Movie.

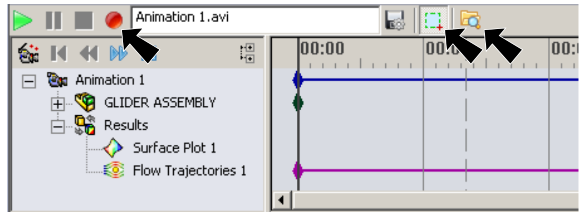
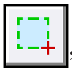
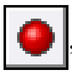


Fig. 37

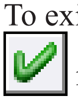
Step 1. Click **Dimetric**  on the Views toolbar.

Step 2. In the Animation control panel turn on **Capture Region** , **Fig. 37**.

Step 3. Drag the red frame capture region in the graphics area to resize and move capture region, **Fig. 38**.

Step 4. Click **Record** , **Fig. 37**.

Step 5. Click **Open Folder**  to view Animation AVI file.

Step 6. To exit, click **OK**  in Animation control panel.

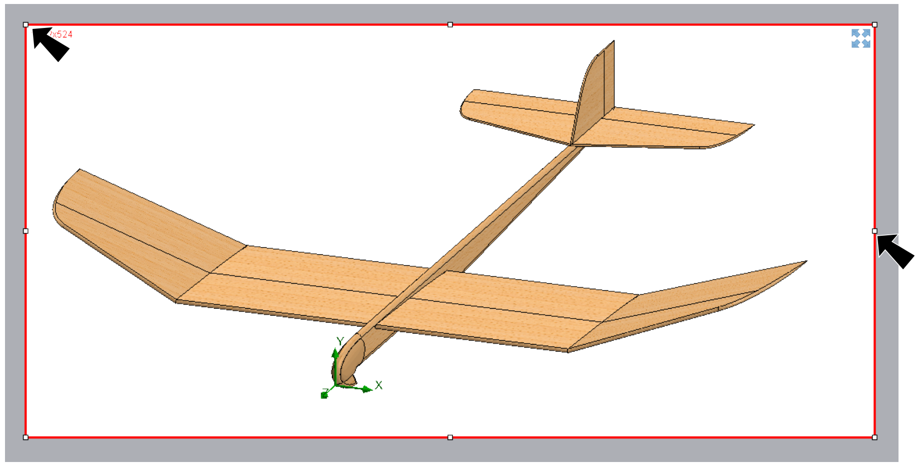


Fig. 38

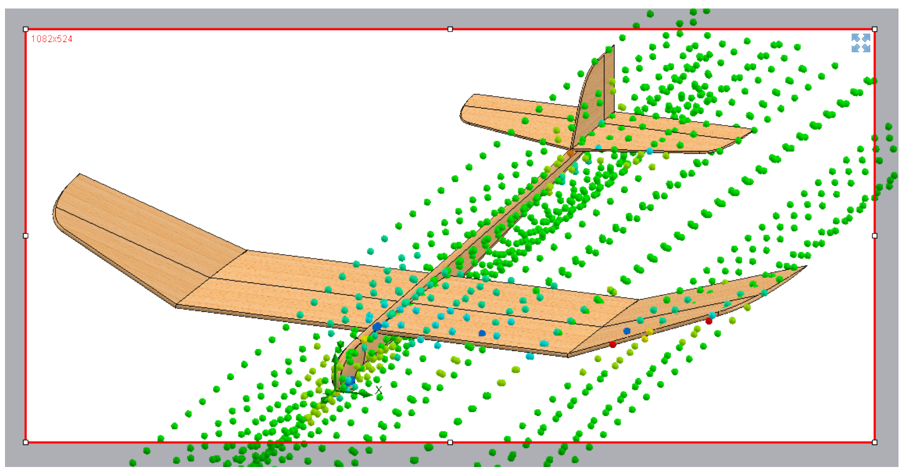


Fig. 39

L. Clone Project 4 Degrees Angle of Attack.

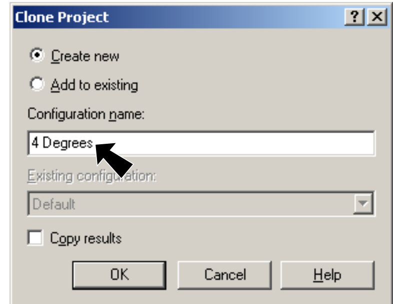


Fig. 40

Step 1. Click **Right** on the Standard Views toolbar. **(Ctrl-4)**

Step 2. Click Flow Simulation Menu > Project > **Clone Project**.

Step 3. In the Clone Project dialog box:
under Configuration name, **Fig. 40**
key-in **4 Degrees**
click OK.

Step 4. Click the **Feature Manager tab** in the Feature Manager, **Fig. 41**.

Step 5. Expand **Mates** in the Feature Manager and click **Angle1**, **Fig. 41**.

Step 6. **Double click** the 2 degrees dimension to modify, **Fig. 42**.

Step 7. In the Dimension Modify box,
click the Configuration drop down arrow and select
This Configuration, **Fig. 43**
key-in **4** for the angle, **Fig. 44**
click OK, **Fig. 45**.

Step 8. Click **Rebuild** **(Ctrl-B)** on the Standard toolbar.

Step 9. Click **No to reset domain**.

Step 10. Click **Yes to reset mesh**.

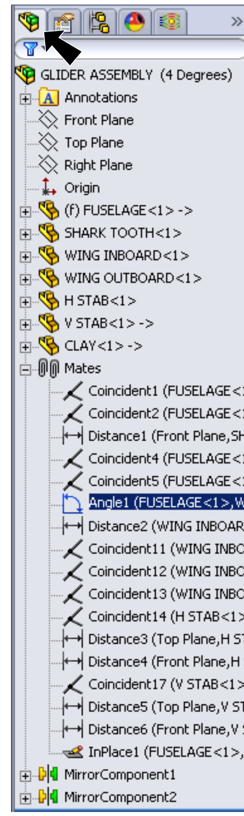


Fig. 41

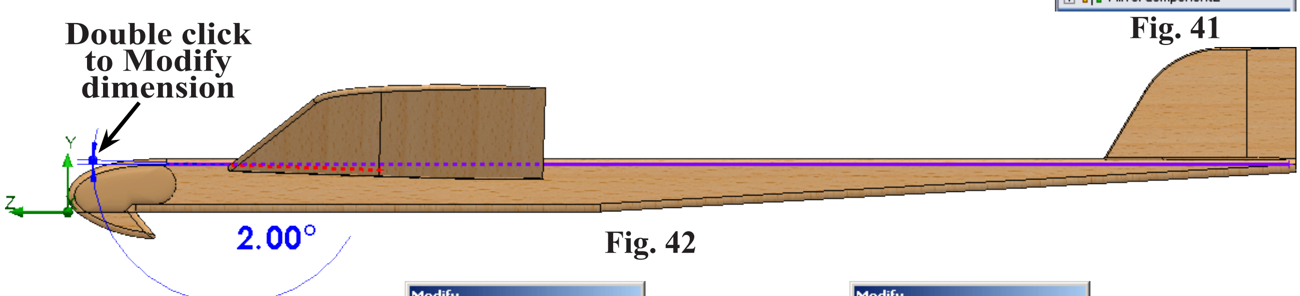


Fig. 42

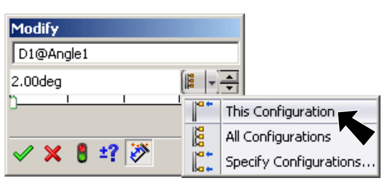


Fig. 43

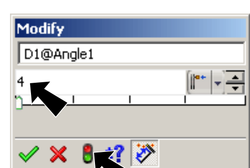


Fig. 44

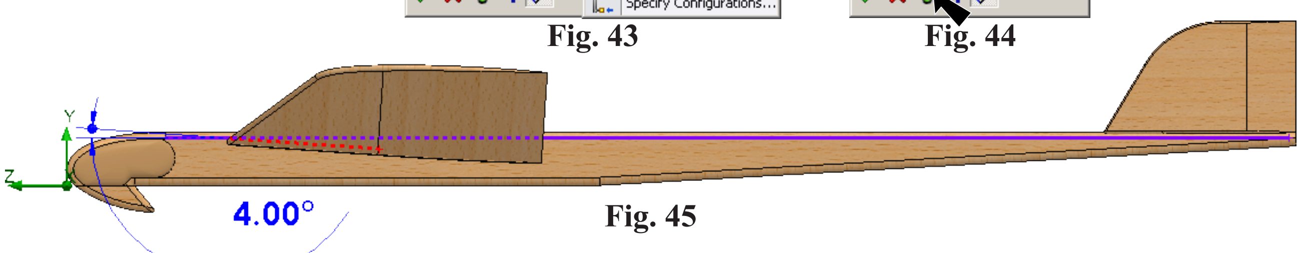


Fig. 45

M. Clone Project 6 Degrees Angle of Attack.

Step 1. Click Flow Simulation Menu > Project > **Clone Project**.

Step 2. In the Clone Project dialog box:
under Configuration name, **Fig. 46**
key-in **6 Degrees**
click OK.

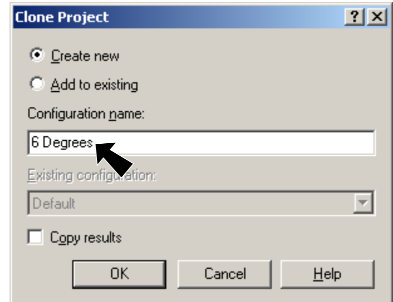


Fig. 46

Step 3. Expand **Mates** in the Feature Manager and click **Angle1**, **Fig. 47**.

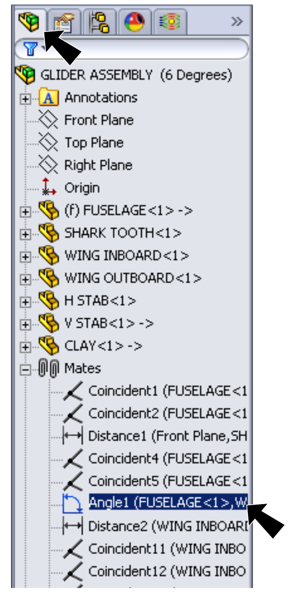


Fig. 47

Step 4. **Double click** the 4 degrees dimension to modify, **Fig. 48**.

Step 5. In the Dimension Modify box,

confirm **This Configuration** is selected, **Fig. 49**
key-in **4** for the angle

click **Rebuild**
click **No to reset domain**
click **Yes to reset mesh**

click OK, **Fig. 50**.

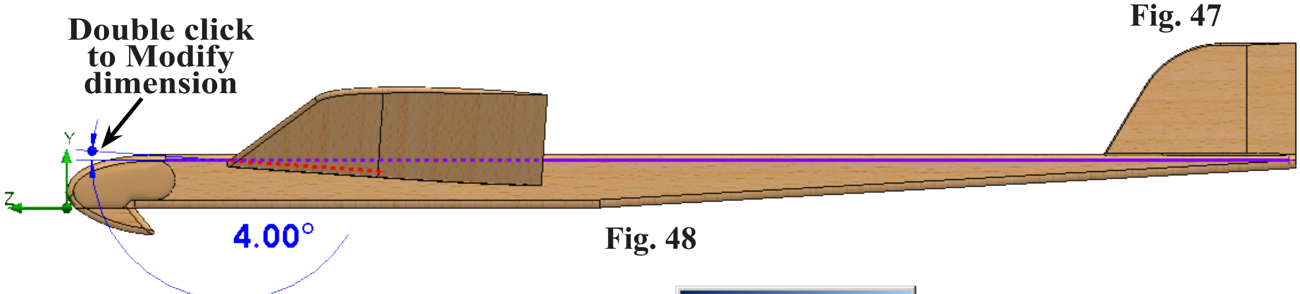


Fig. 48

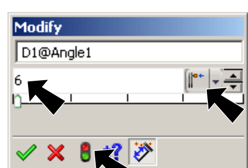


Fig. 49

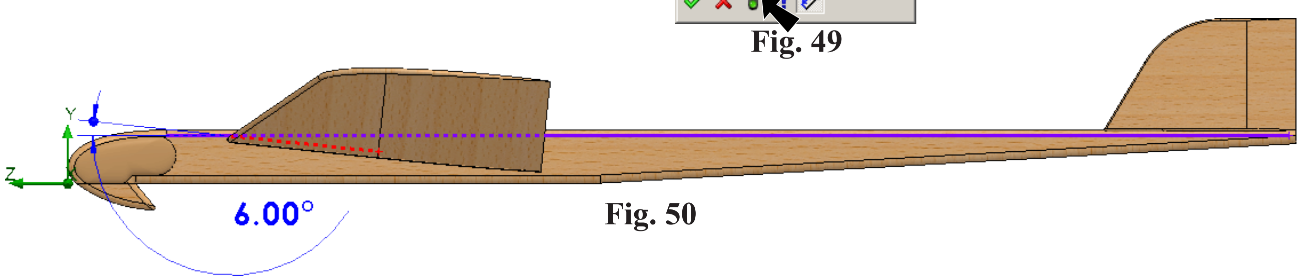


Fig. 50

N. Batch Run Analysis.

Step 1. Click Flow Simulation Menu > Solve > **Batch Run**.

Step 2. In the Batch Run dialog box:
check **All projects Solve** check box
click **Run**
Fig. 51.
The 2 Degrees Solver will be displayed and then 4 and 6 Degrees Solvers will be calculated.
The Solvers create huge subfolders.
If you are working off a network with a file saving allotment you might want to consider deleting the subfolders once this project is completed.

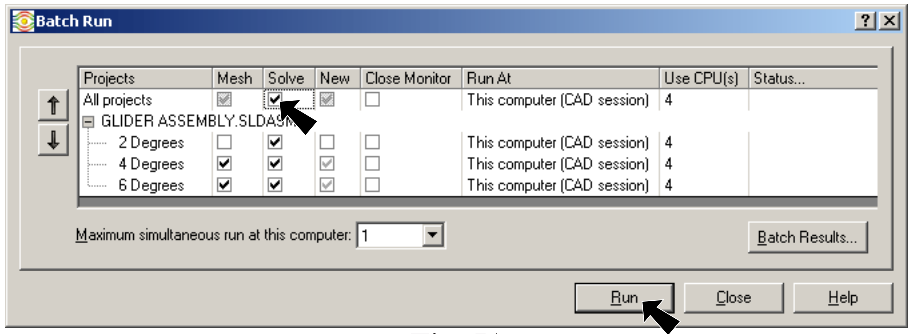


Fig. 51

Step 3. Which Angle of Attach creates the most Lift?
What happens to the Drag?
Try 8 and 10 degrees.
What happens between 8 and 10 degrees?

O. Access Configurations.

Step 1. To access the configurations, click **Configurations Manger tab**  in the Feature Manager, **Fig. 52**.

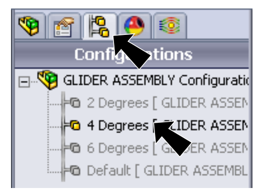


Fig. 52

Step 2. **Double click** a configuration to select it, **Fig. 52**.

Step 3. Click Flow Simulation Menu > Results > **Load/Unload Results**.

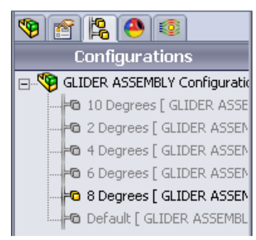


Fig. 53