

Airplane Flow Sim

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.

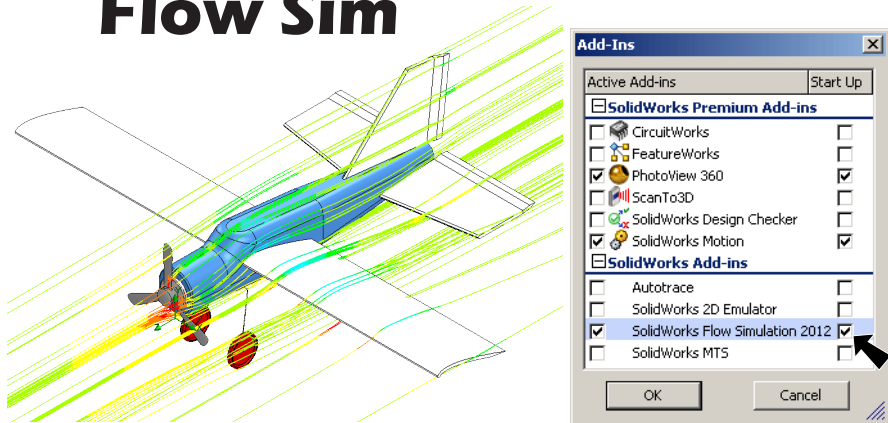
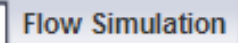

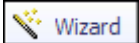


Fig. 1

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.

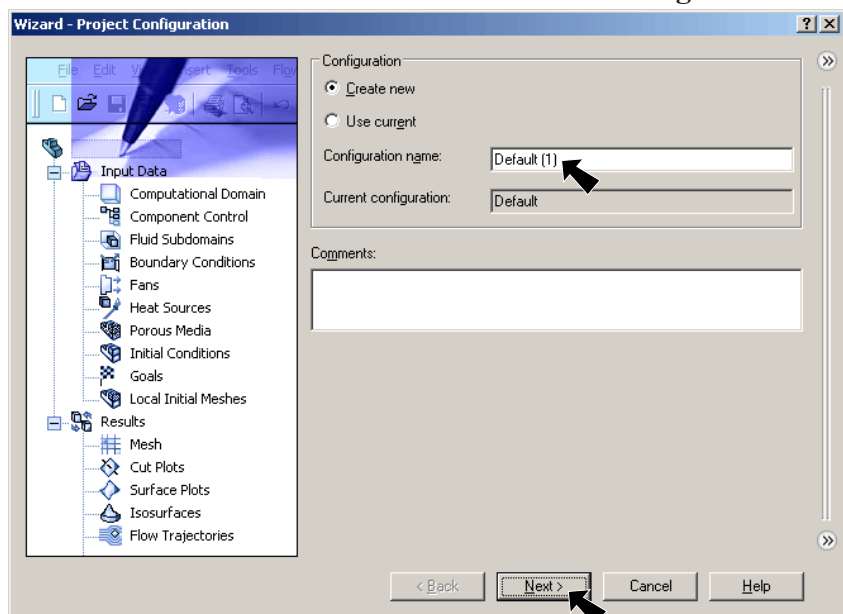


Fig. 2

- Step 3. Use **Default** for Configuration name, Fig. 2.
- Step 4. Click Next.
- Step 5. **Unit System:** Select **SI (m-k-g-s)** for Unit system, Fig. 3.
- Step 6. Under **Parameter**, set **Velocity** units to **Mile/hour**, Fig. 3.

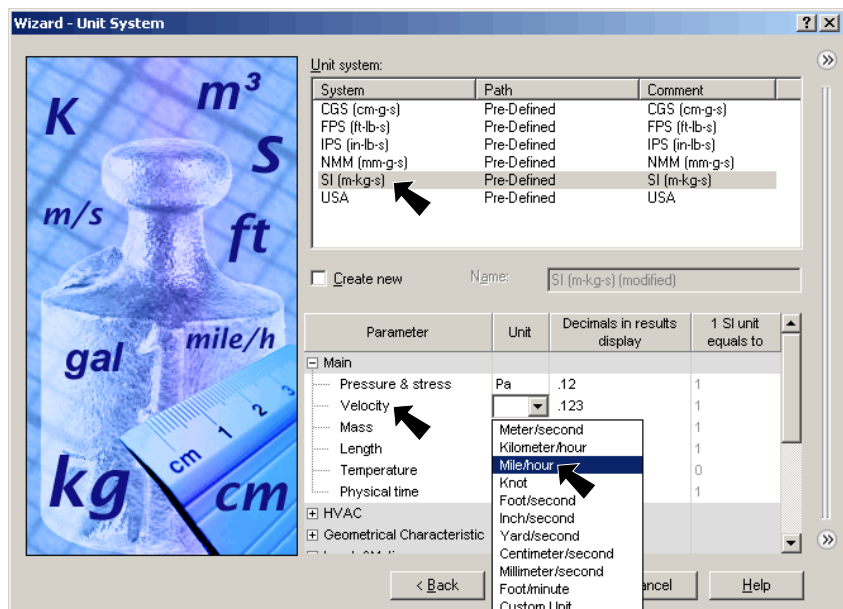


Fig. 3

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

Step 16. Click Next.

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

Step 18. Click Next.

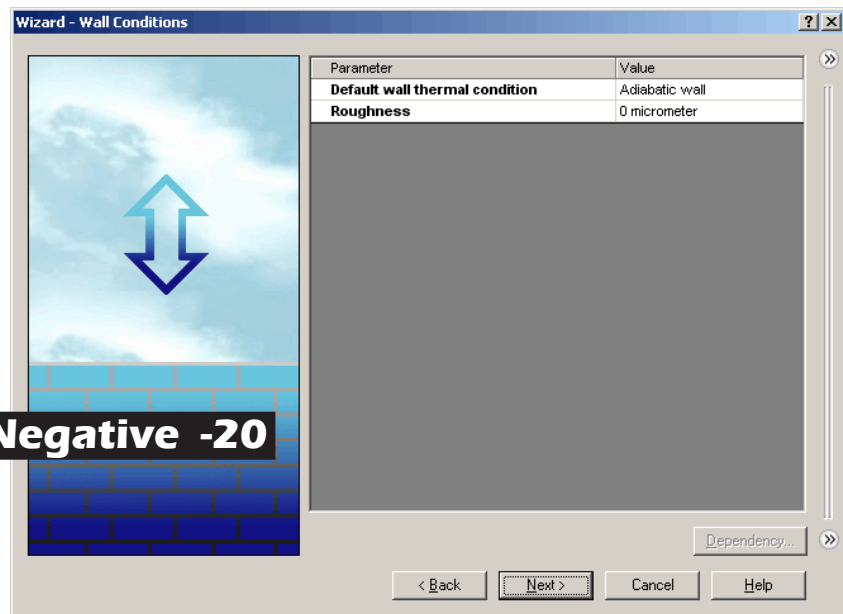


Fig. 7

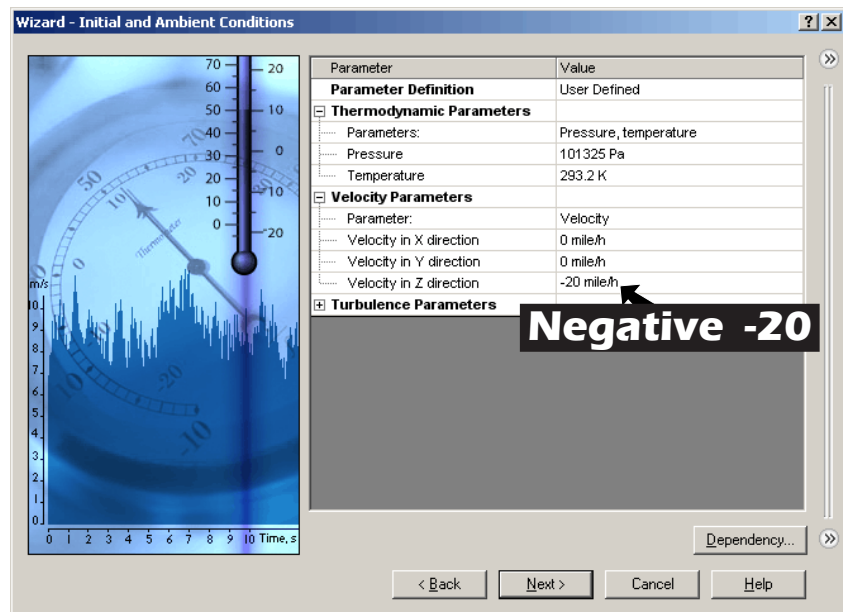


Fig. 8

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.

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

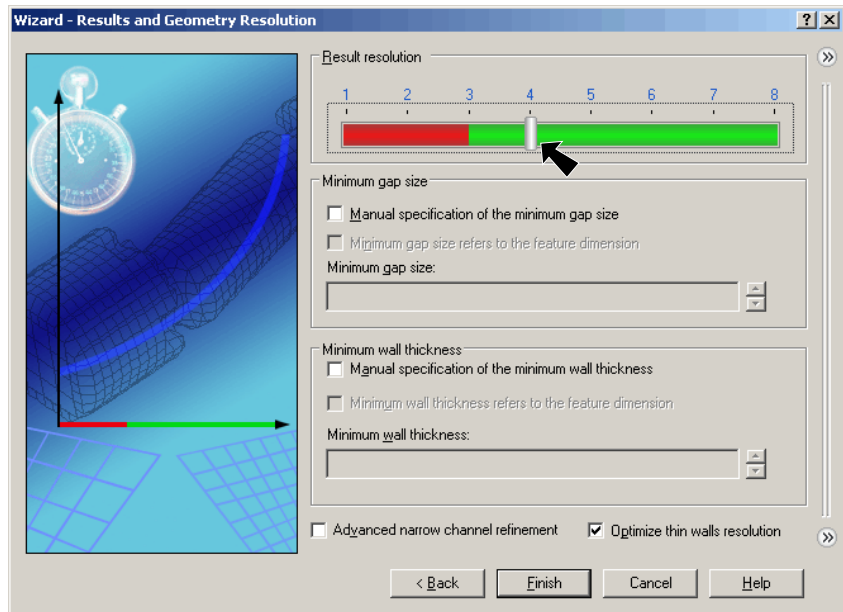


Fig. 9

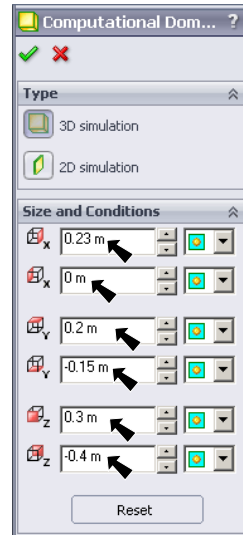


Fig. 10

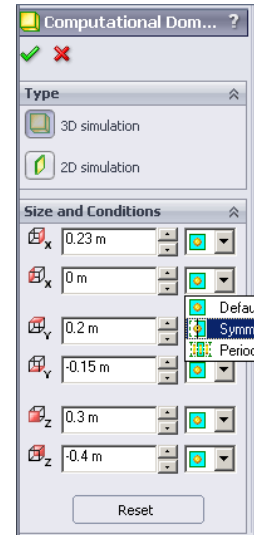


Fig. 11

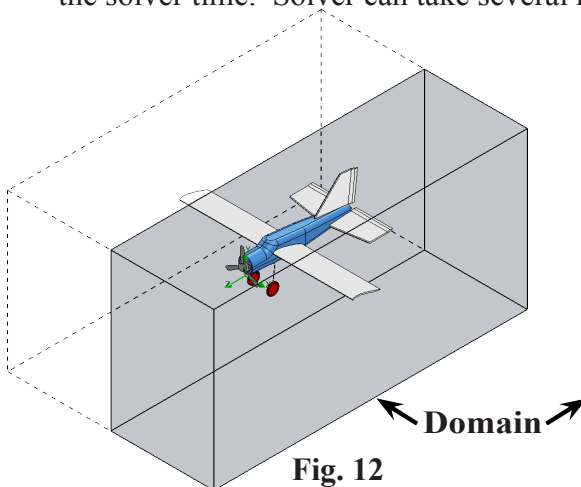


Fig. 12

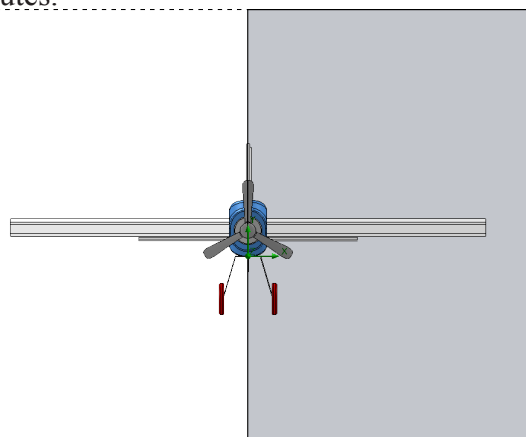


Fig. 13

D. Insert Global Goals.

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

Step 2. Click the **Flow Simulation** tab  in the Feature Manager, **Fig. 14**.

Step 3. Click Flow Simulation Menu > Insert > **Global Goals**.

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**.

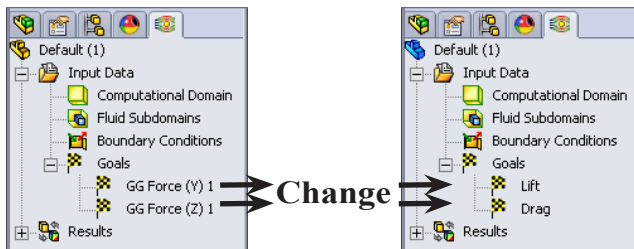


Fig. 15

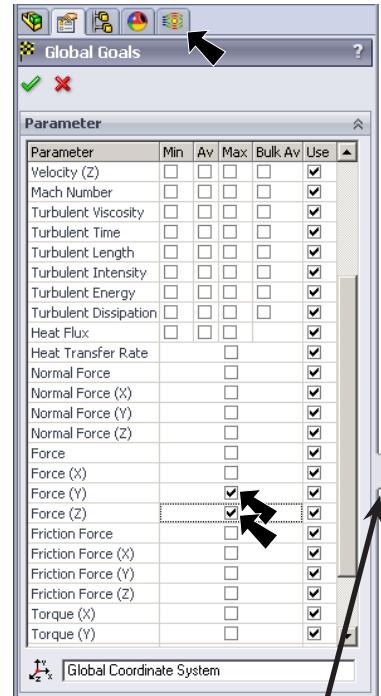




Fig. 14
Drag right

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**.

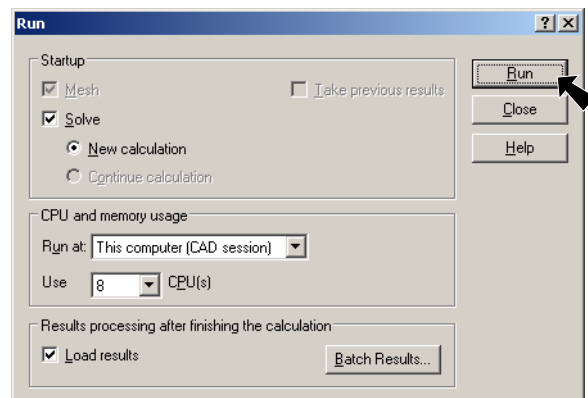


Fig. 16

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 airplane design and computer.

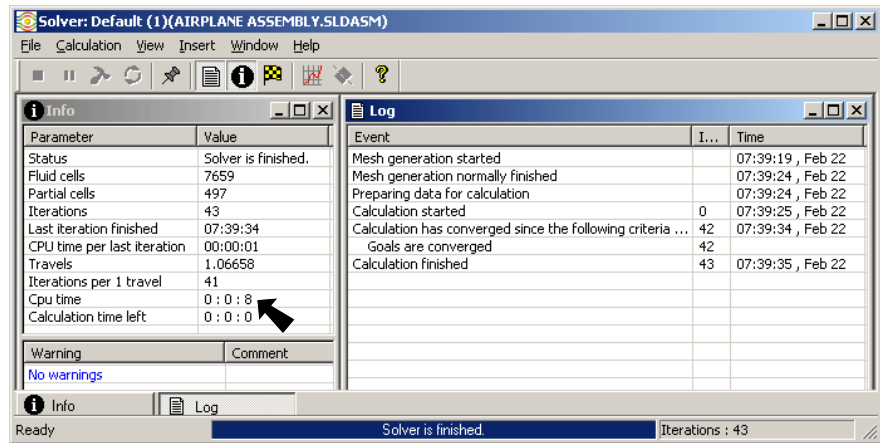


Fig. 17

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

Step 3. Click **Add All** and OK in Add/Remove Goals, **Fig. 18**. Our Lift was **31.39**, **Fig. 19**.

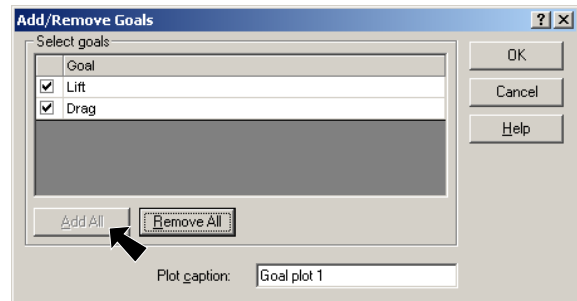


Fig. 18

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

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

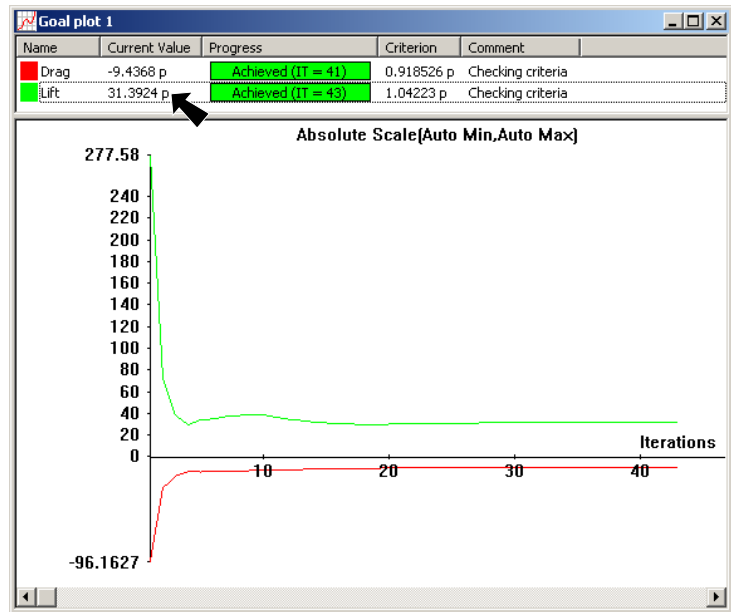


Fig. 19

G. Surface Plots.

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

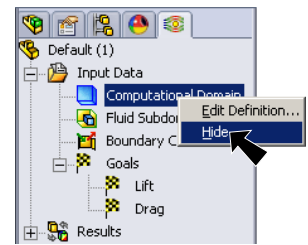




Fig. 20

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 **101350**

Minimum  pressure **101280**

click OK .

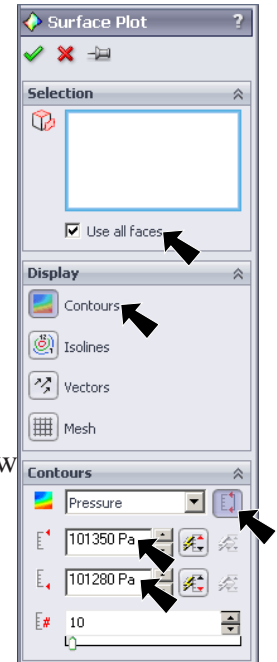


Fig. 21

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**.

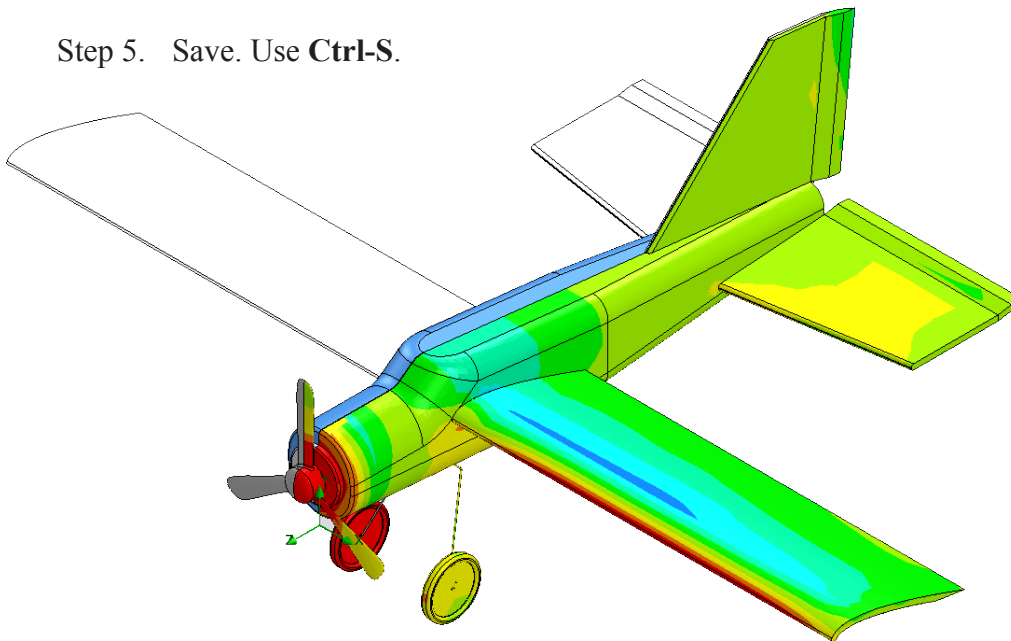


Fig. 22

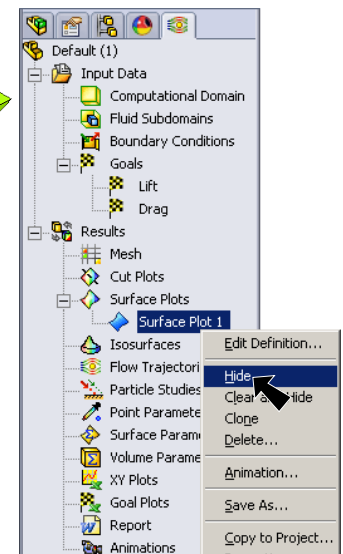


Fig. 23

H. Flow Trajectories.


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. 24

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

Step 3. In the Flow Trajectories Property Manager:
under **Starting Points**, **Fig. 25**

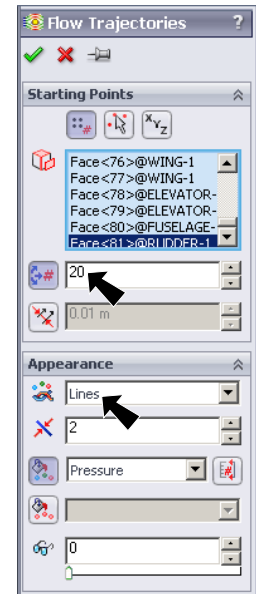


Fig. 25

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, Wing, H Stab and V Stab. Click all remains surfaces that have not been selected on the right side of the airplane. Rotate view to select bottom and rear faces, **Fig 27**.

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


under **Options**

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

click **OK** , **Fig. 28**.

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

Step 5. **Turn off Filter**

Faces  (X) on the **Selection Filter toolbar** at the bottom of the display, **Fig. 24**. **Click faces** **Fig. 27**

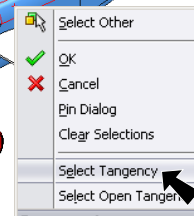
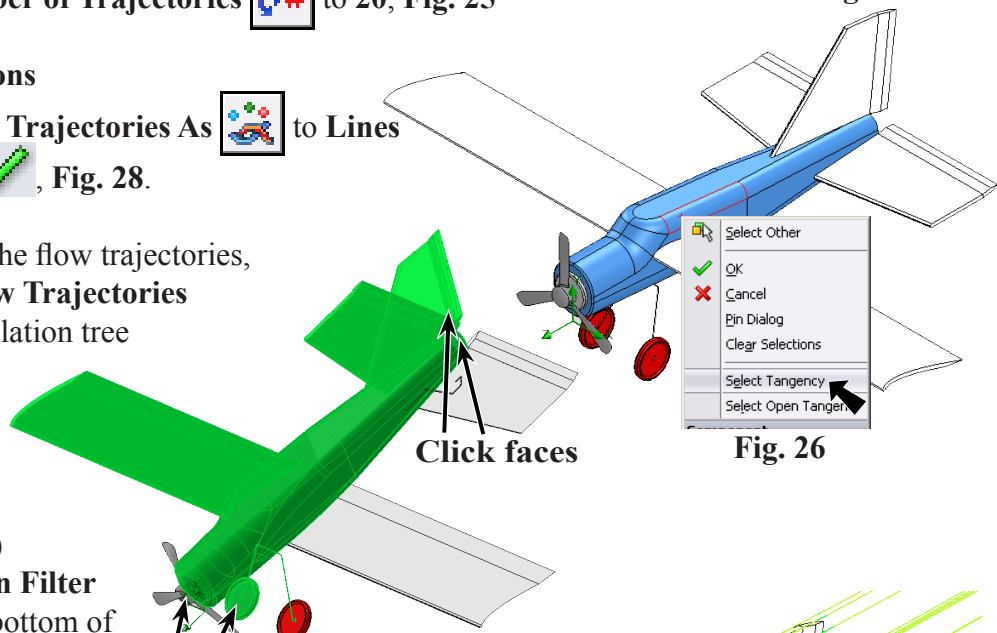


Fig. 26

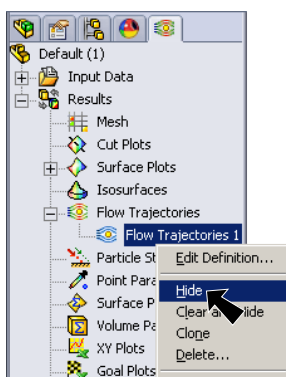


Fig. 29

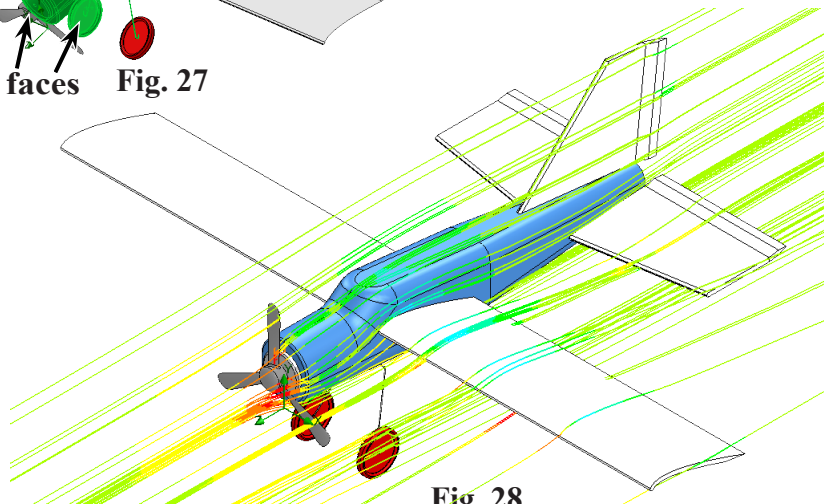


Fig. 28

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**.

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.

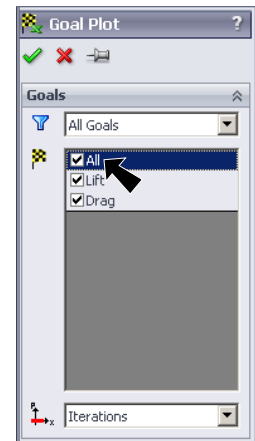


Fig. 30

Microsoft Excel - goals1

AIRPLANE ASSEMBLY.SLDASM [Default (1)]									
Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Lift	[p]	31.39242274	31.26393547	30.59019793	31.4948201	100	Yes	0.904622164	1.04222593
Drag	[p]	-9.436803149	-9.571754199	-9.964552409	-9.436803149	100	Yes	0.52774926	0.918525792

Iterations: 43
Analysis interval: 21

Fig. 31

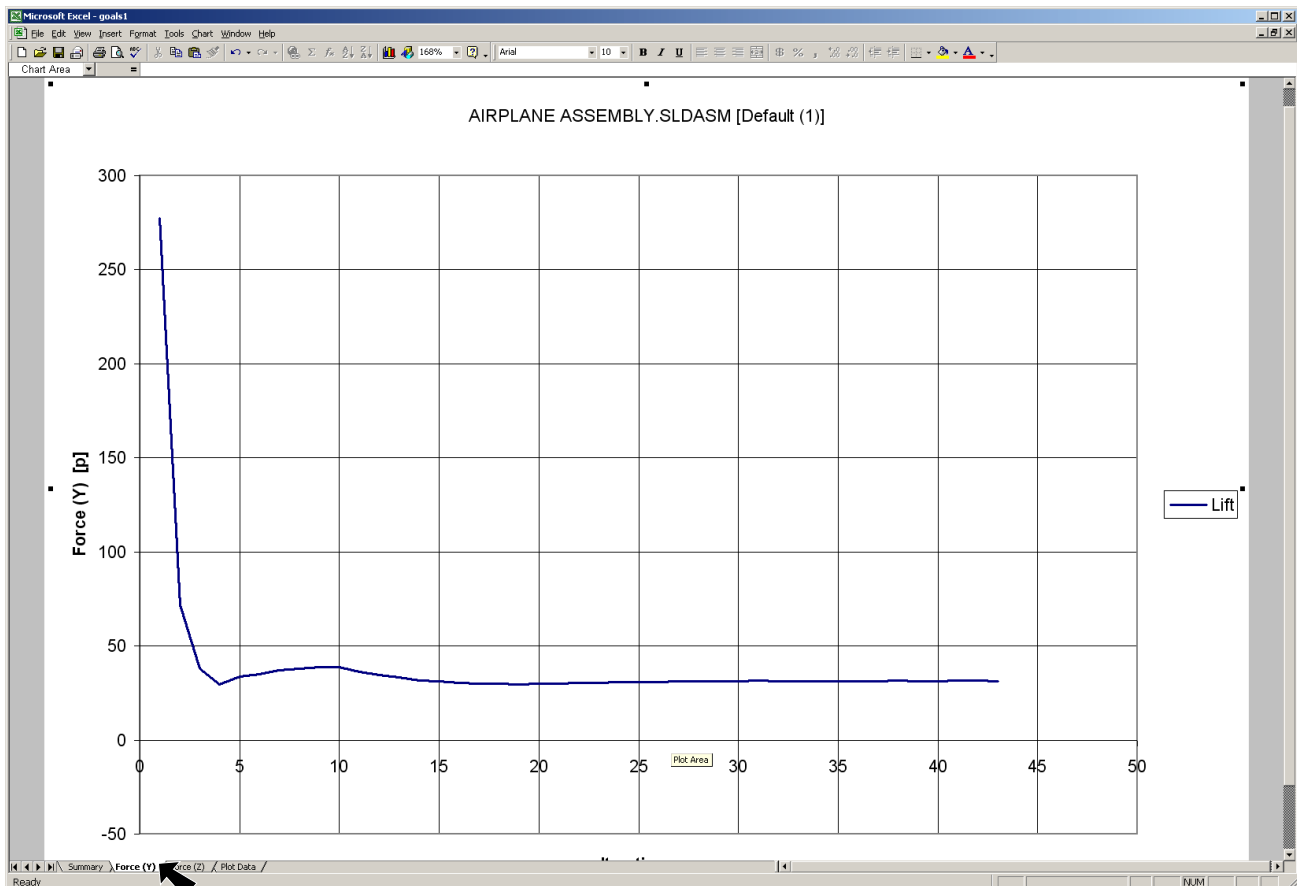




Fig. 32

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.

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:

under **Appearance**, Fig. 35

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

set **Cross Size**  to .003

click **OK** .

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.

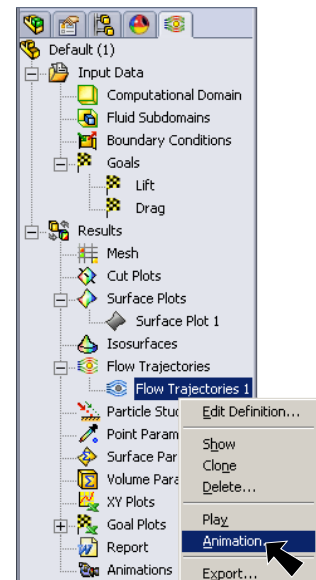


Fig. 33

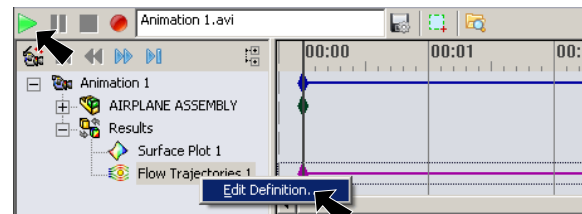


Fig. 34

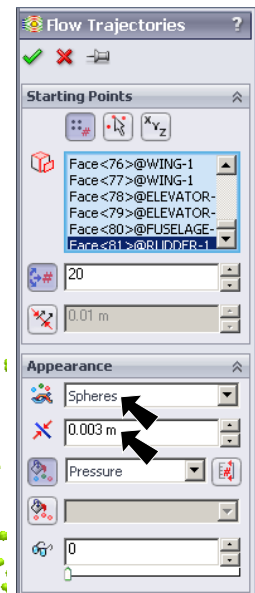


Fig. 35

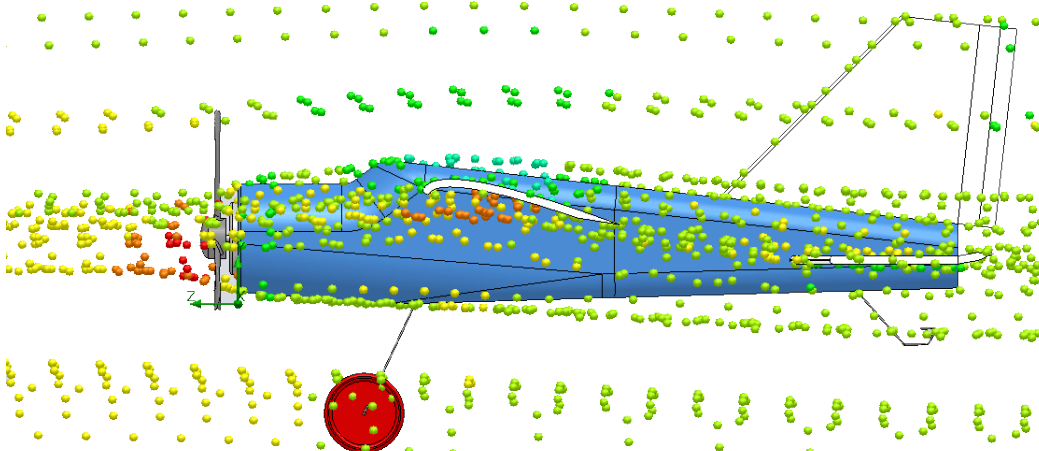



Fig. 36

K. Create Animation Movie.

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

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

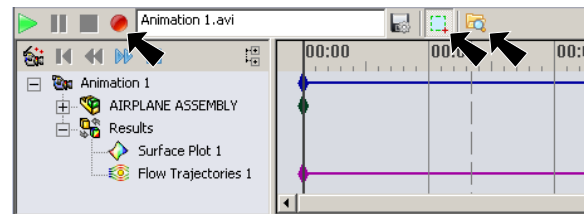




Fig. 37

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.

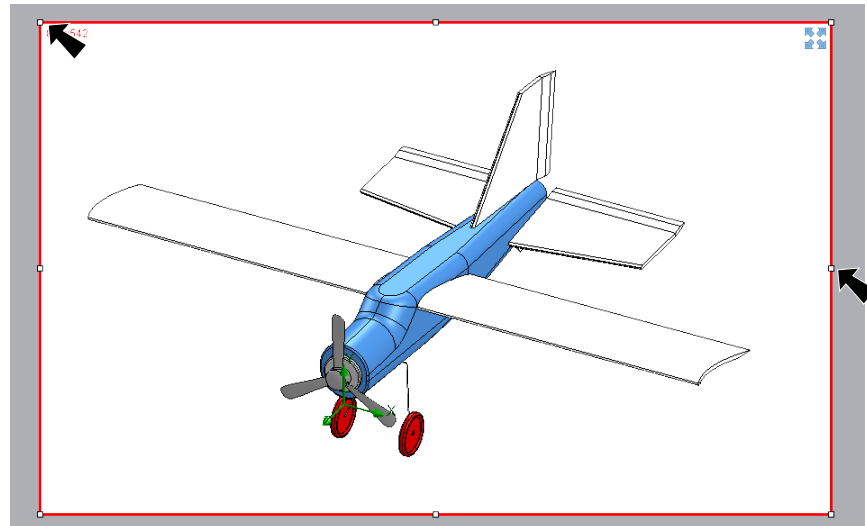


Fig. 38

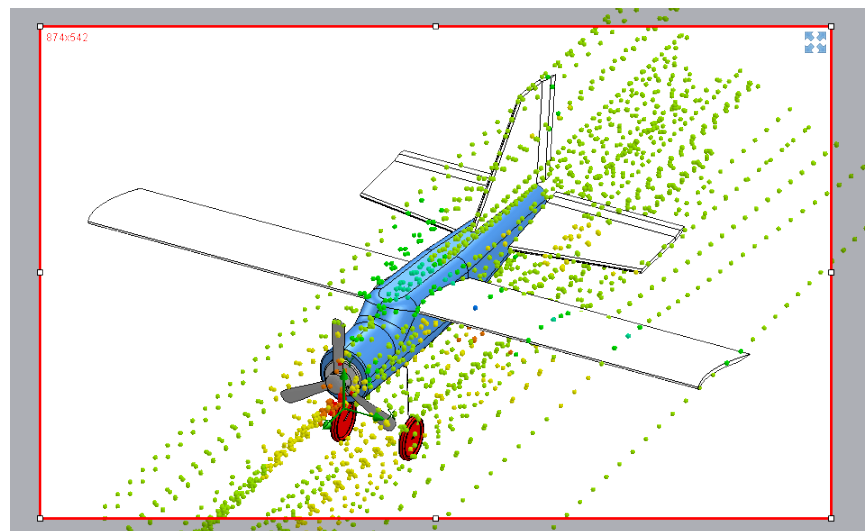







Fig. 39

L. Modify Angle of Attack.

- Step 1. Click **Right**  on the Standard Views toolbar. (Ctrl-4)
- Step 2. Click the **Feature Manager** tab  in the Feature Manager, Fig. 40.
- Step 3. Expand **WING** and expand **Mates in AIRPLANE ASSEMBLY**, Fig. 40.
- Step 4. Click **Angle1** in the Feature Manager and click **Edit Feature**  from the Content toolbar, Fig. 40.
- Step 5. In the Property Manager set,
 **Angle1** to 13, Fig. 41
 click OK , Fig. 43.

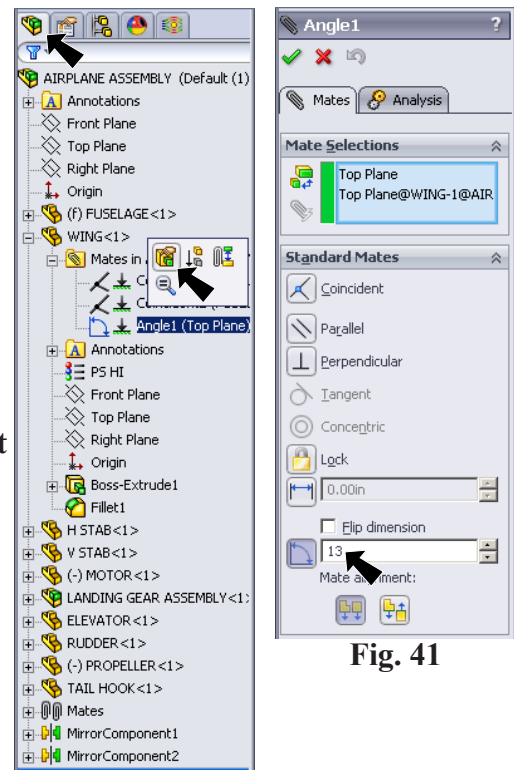





Fig. 40

- Step 6. Click **No** to reset domain.
- Step 7. Click **Yes** to reset mesh. Click OK  again.

M. Run Analysis.

- Step 1. Click **Flow Simulation**  on the Flow Simulation toolbar and **Run**  from the menu.
- Step 2. Click **Run** in the Run dialog box.
- Step 3. What is the new Lift? What happened to the drag?

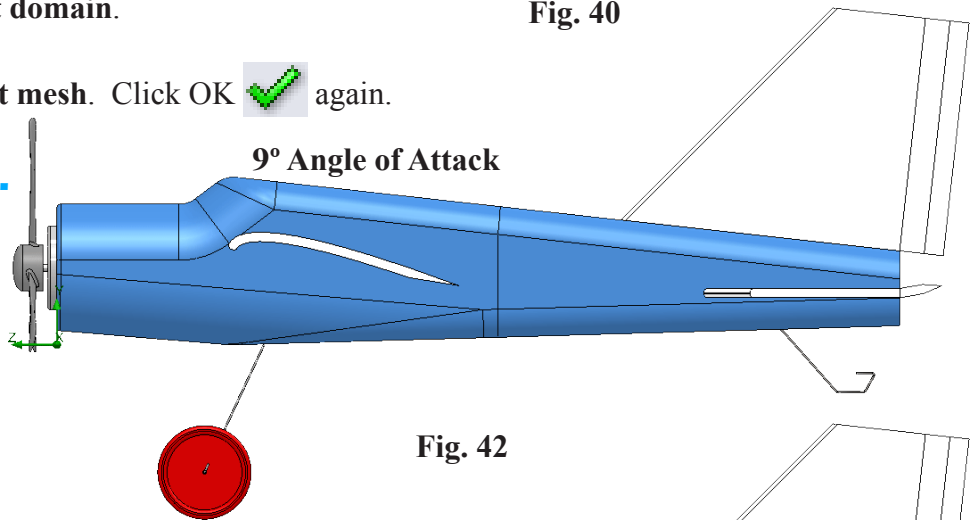


Fig. 42

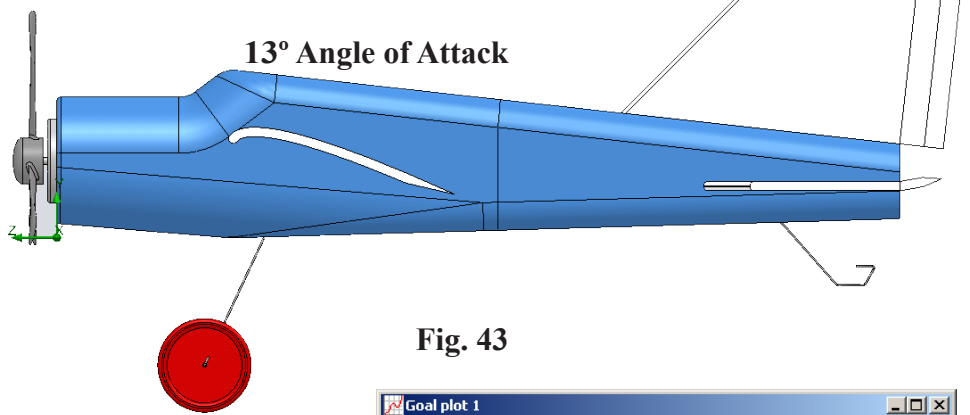
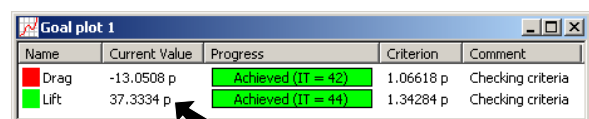


Fig. 43



Name	Current Value	Progress	Criterion	Comment
Drag	-13.0508 p	Achieved (IT = 42)	1.06618 p	Checking criteria
Lift	37.3334 p	Achieved (IT = 44)	1.34284 p	Checking criteria

Fig. 44