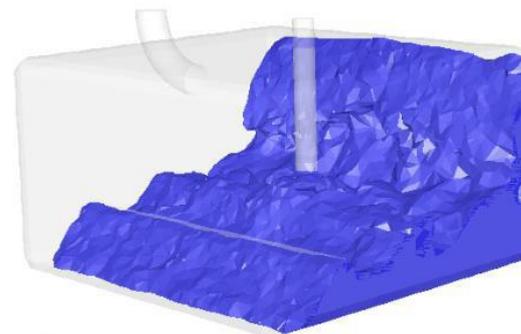


油箱晃动仿真流程

CFD-Team



所属: IDAJ

目录

■ 概述

■ 模型简介

■ 计算设定

- ◆ 软件启动

- ◆ 网格处理

- ◆ 物理模型设定

- ◆ 求解器设定

- ◆ 后处理设定

 - 计算输出设定

 - 瞬态计算图片输出

- ◆ 提交计算

■ 总结

概述

- 随着汽车工业的发展，乘客对汽车性能、舒适性等要求越来越高。安全性、噪音等成为汽车行业关注的要点
- 车辆在颠簸路面行驶、启停、拐弯时，其燃油箱内部的燃油发生晃动，可能出现燃油箱结构损坏、吸油泵吸油不畅和噪音等问题
- 研究燃油箱内部油液晃动，避免由于燃油箱晃动引起的问题，是一个迫在眉睫的课题
- **ANSYS Fluent**具有快速、高精度VOF求解器，可快速准确分析和解决燃油箱晃动带来的问题
- 本文基于**ANSYS Fluent 17.0**，介绍燃油箱晃动时，燃油液面位置状态变化的仿真流程

目录

■ 概述

■ 模型简介

■ 计算设定

- ◆ 软件启动

- ◆ 网格处理

- ◆ 物理模型设定

- ◆ 求解器设定

- ◆ 后处理设定

 - 计算输出设定

 - 瞬态计算图片输出

- ◆ 提交计算

■ 总结

模型简介-几何模型

■ 几何模型：

- ◆ 油箱包含：注油管、吸油管、油箱壳体和防浪板等部件
- ◆ 本例针对图1有防浪板模型，车辆加速过程进行设定分析

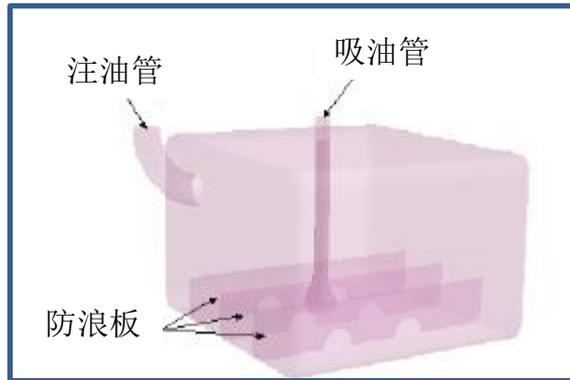


图1 有防浪板模型

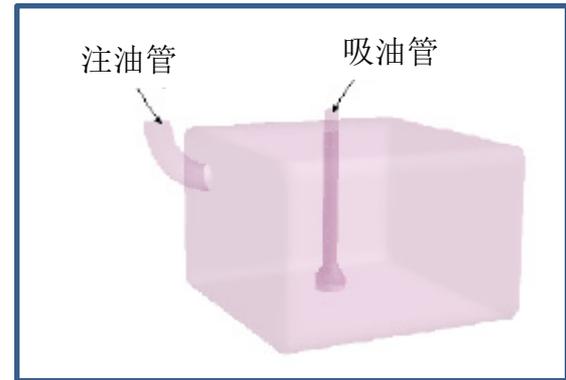
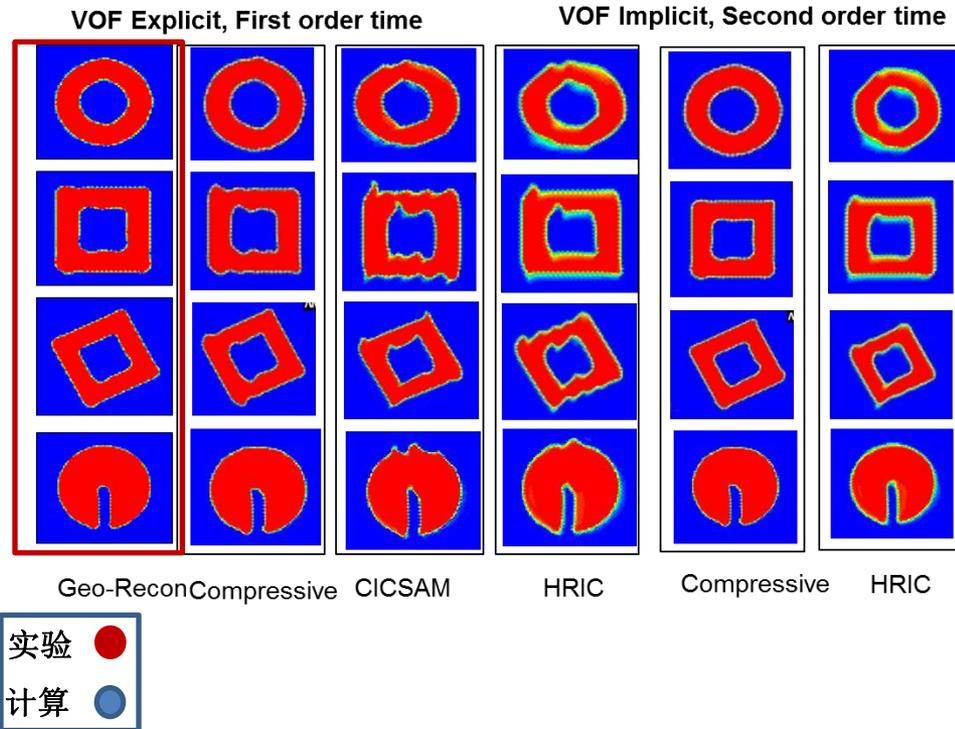
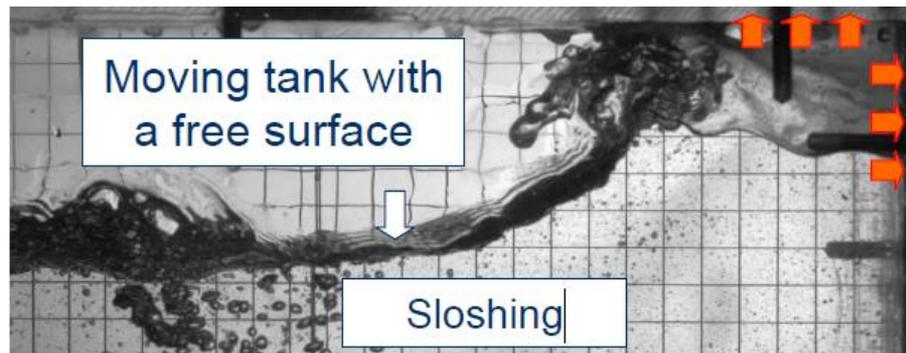


图2 无防浪板模型

模型简介-物理模型

■ Fluent VOF模型

- ◆ 高精度VOF模型，可以清晰捕捉液面



注：图片引用自《Best Practices for Sloshing/Filling Applications》

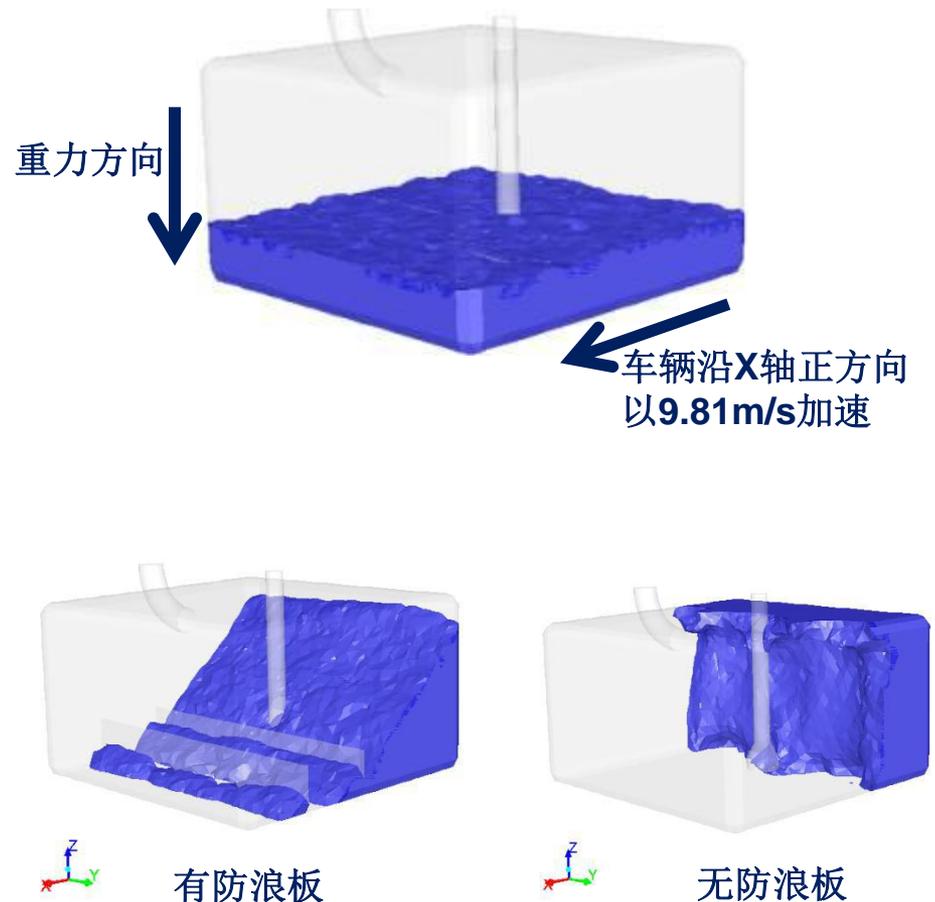
模型简介-物理模型

■ 油箱晃动主要模型：

- ◆ 瞬态求解器
- ◆ VOF模型

■ 模拟物理现象

- ◆ 油箱沿X轴正方向加速： 9.81m/s
- ◆ 重力加速度沿Z轴负方向： 9.81m/s
- ◆ 计算物理时间： 1.25s



目录

■ 概述

■ 模型简介

■ 计算设定

- ◆ 软件启动

- ◆ 网格处理

- ◆ 物理模型设定

- ◆ 求解器设定

- ◆ 后处理设定

 - 计算输出设定

 - 瞬态计算图片输出

- ◆ 提交计算

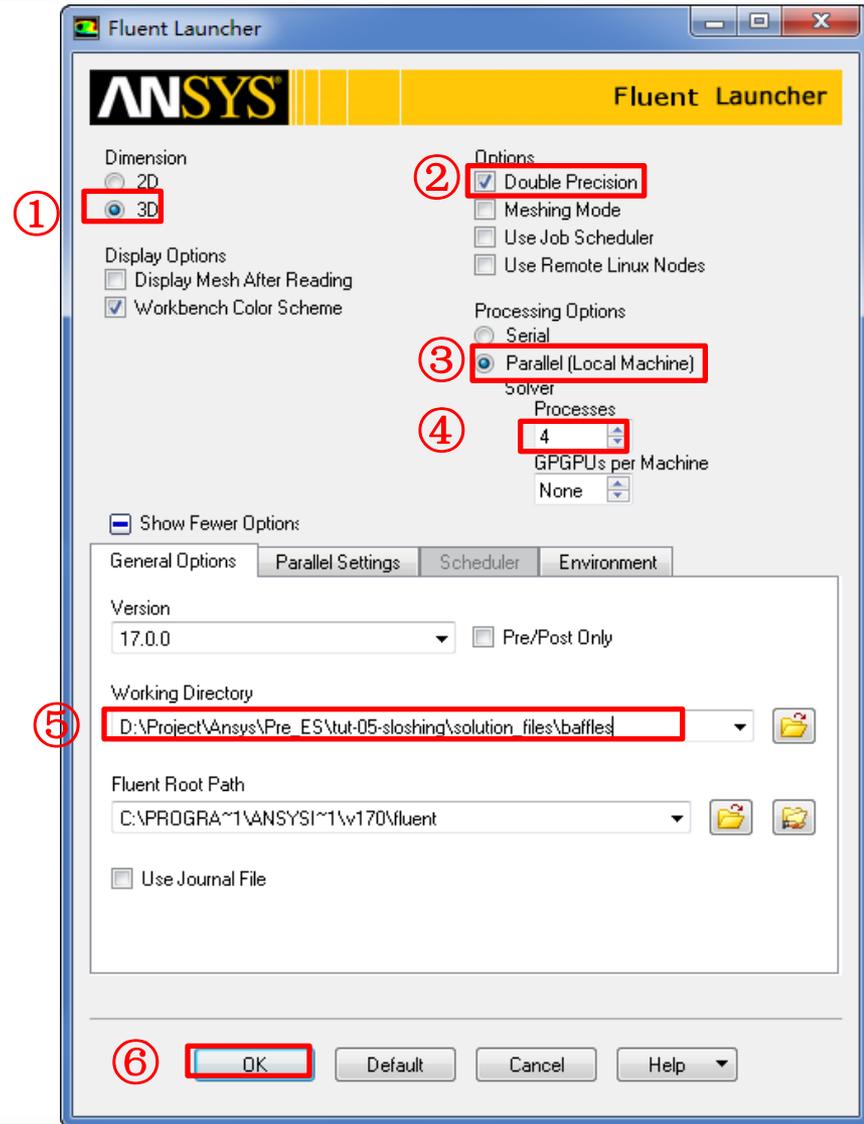
■ 总结

计算设定

■ 启动软件

◆ 双击Fluent图标，开启Fluent

- 1) 选择三维求解器
- 2) 双精度求解器
- 3) 并行计算
- 4) 根据计算机性能和求解规模输入求解用CPU数量
- 5) 工作路径

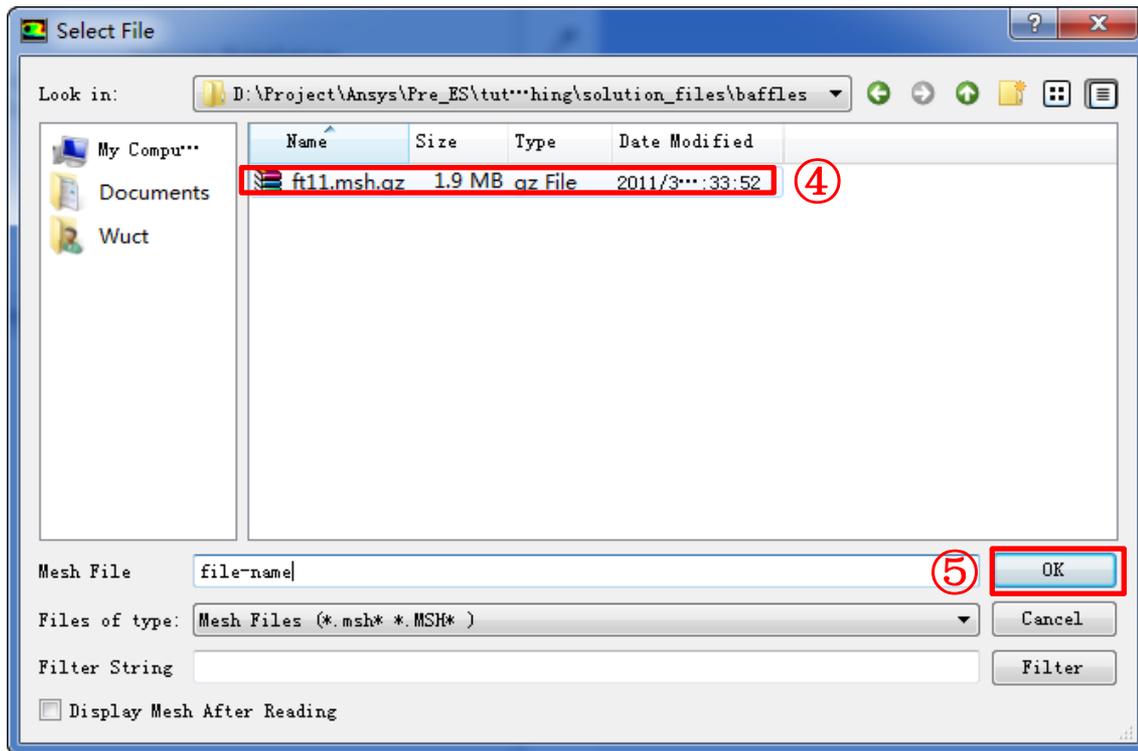
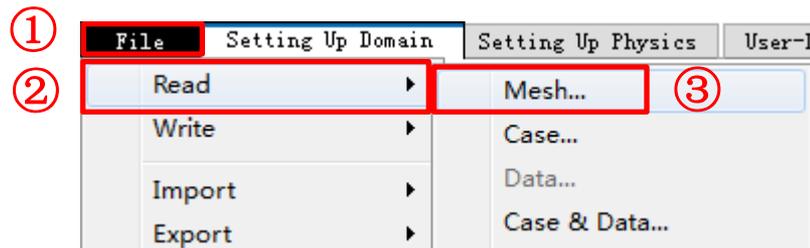
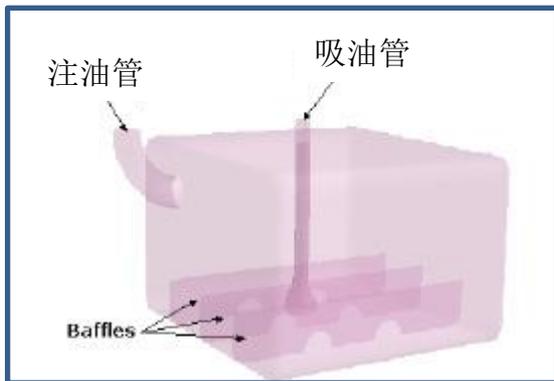


计算设定

■ 网格处理

◆ 读取体网格

➤ 导入油箱体网格



计算设定

网格处理

检查网格:

- 1) 网格尺寸: Minimum volume > 0
- 2) 网格质量: Max Ortho Skew < 0.9

调整网格尺寸单位

Domain Extents:

x-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e+01
 y-coordinate: min (m) = -1.240000e+01, max (m) = 5.000000e+01
 z-coordinate: min (m) = 0.000000e+00, max (m) = 3.400000e+01

Volume statistics:

minimum volume (m3): 7.524521e-03
 maximum volume (m3): 3.621122e+00
 total volume (m3): 7.471332e+04

Face area statistics:

minimum face area (m2): 5.599807e-02
 maximum face area (m2): 5.282123e+00

Checking mesh.....
 Done.

⑤

检查网格全局尺寸是否在合理范围内: 建模时以cm为单位, 模型导入fluent后默认尺寸单位为m...需要调整尺寸单位

Mesh Quality:

Minimum Orthogonal Quality = 2.79773e-02
 (Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)

Maximum Ortho Skew = 7.51369e-01
 (Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)

Maximum Aspect Ratio = 2.52383e+01

Scale Mesh

Domain Extents

Xmin (m)	0	Xmax (m)	50
Ymin (m)	-12.4	Ymax (m)	50
Zmin (m)	0	Zmax (m)	34

Scaling

Convert Units ⑥
 Specify Scaling Factors

Mesh Was Created In

cm ⑦

Scaling Factors

X	0.01
Y	0.01
Z	0.01

⑧

⑨

执行步骤⑧

Scale Mesh

Domain Extents

Xmin (m)	0	Xmax (m)	0.05
Ymin (m)	-0.0124	Ymax (m)	0.05
Zmin (m)	0	Zmax (m)	0.034

Tree

- Setup
 - General ①
 - Models
 - Materials
 - Cell Zone Conditions
 - Boundary Conditions
 - Dynamic Mesh
 - Reference Values
- Solution
 - Solution Methods
 - Solution Controls
 - Monitors
 - Report Definitions
 - Report Files
 - Report Plots
 - Solution Initialization
 - Calculation Activities
 - Run Calculation
- Results
 - Graphics
 - Animations
 - Plots
 - Reports
 - Parameters & Customization

Task Page

General

Mesh

③ ② ④

⑤

Solver

Type

Pressure-Based
 Density-Based

Velocity Formulation

Absolute
 Relative

Time

Steady
 Transient

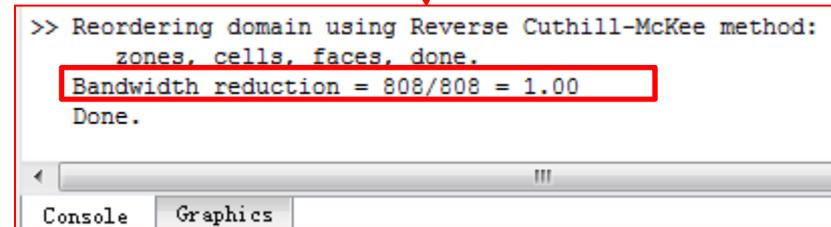
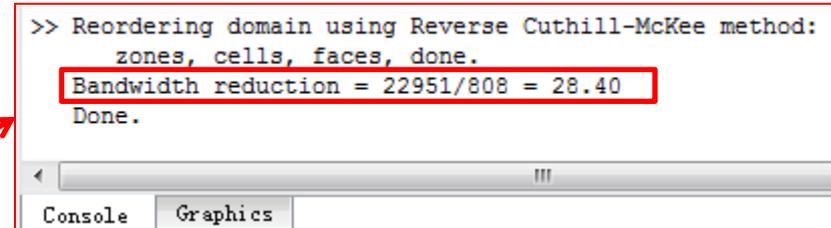
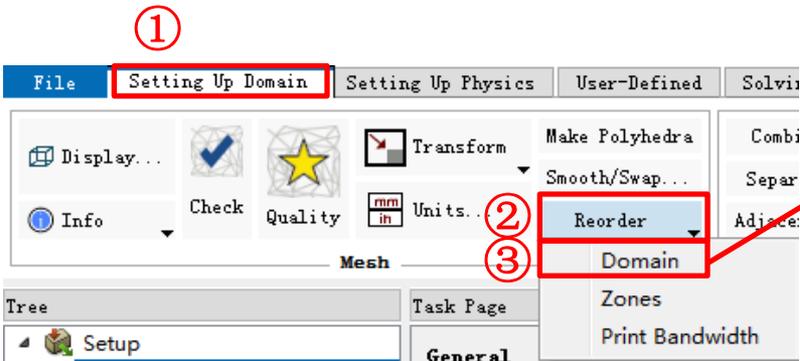
Gravity

计算设定

■ 网格处理

◆ 整理计算域网格

- 重新分配网格节点在内存中的分配，提高计算速度
- 重复执行至 $\text{Bandwidth reduction} \leq 1$



计算设定

物理模型设定

1) Pressure-Based: 压力基求解器

2) Transient: 瞬态求解器

3) Gravity: 加速度

➤ X: 车运动加速度方向

➤ Z: 重力加速度方向

The screenshot shows the ANSYS Fluent software interface. The left pane displays the 'Tree' view with the 'Setup' folder expanded to 'General'. The right pane shows the 'Task Page' for 'General' settings. Four red boxes with numbers 1 through 4 highlight specific settings:

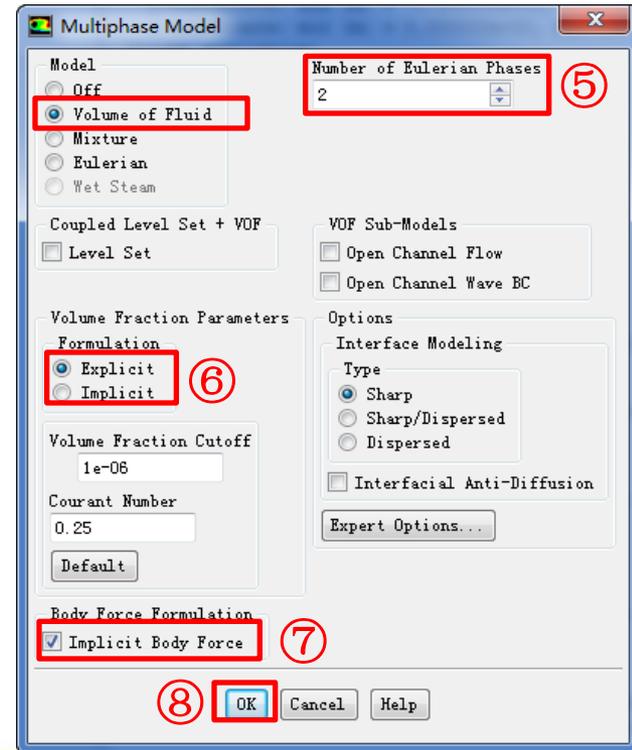
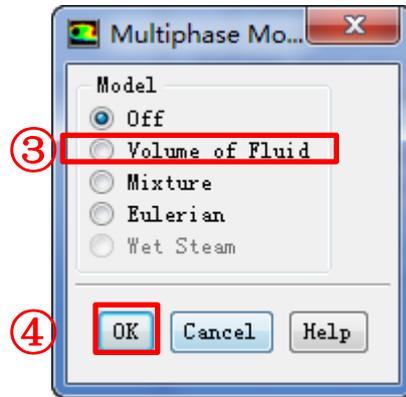
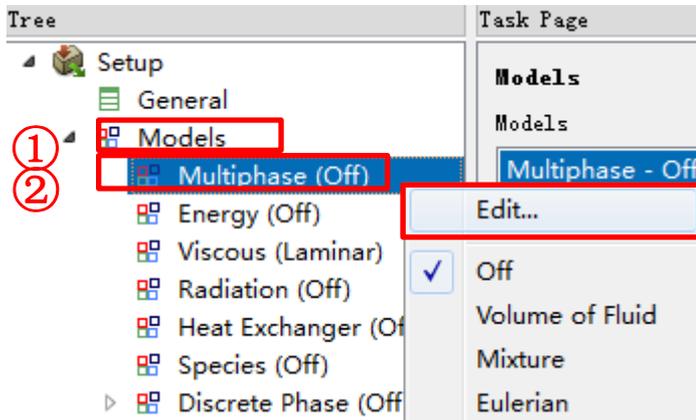
- ①: Solver Type set to **Pressure-Based**.
- ②: Time set to **Transient**.
- ③: **Gravity** checkbox checked.
- ④: Gravitational Acceleration values: X (m/s²) = -9.81, Y (m/s²) = 0, Z (m/s²) = -9.81.

计算设定

■ 物理模型设定

◆ VOF模型选择

- 1) 右键点击Multiphase->edit
- 2) 选择VOF模型: Volume of Fluid
- 3) Number of Eulerian Phases: 两相流
- 4) Explicit: 显式求解
- 5) Implicit Body Force: 体积力



计算设定

物理模型设定

物性设定

创建液相: kerosene-liquid (煤油)

① 右键点击Fluid

② New...

③ Fluent Database...

④ kerosene-liquid (c12h23<|>)

⑤ Copy

⑥ Close

⑦ Change/Create

⑧ Close

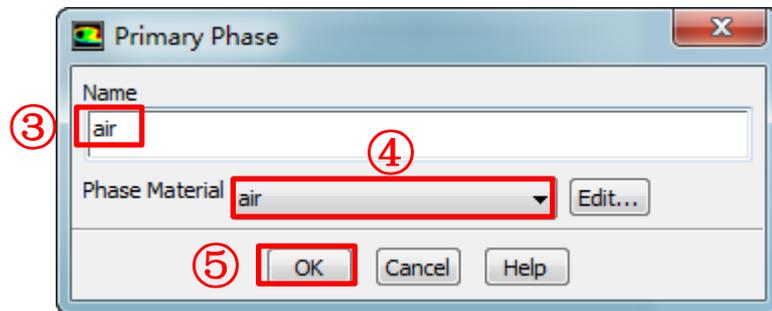
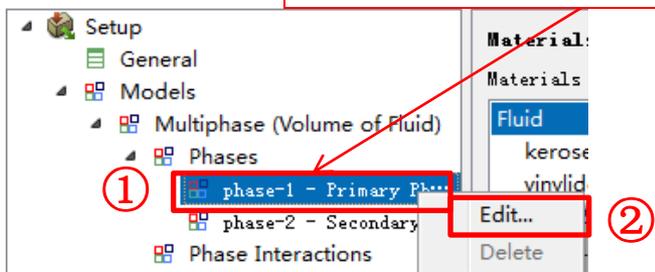
Property	Value
Cp (Specific Heat) (j/kg-k)	piecewise-polynomial
Molecular Weight (kg/kgmol)	constant
Standard State Enthalpy (j/kgmol)	constant
Standard State Entropy (j/kgmol-k)	constant

计算设定

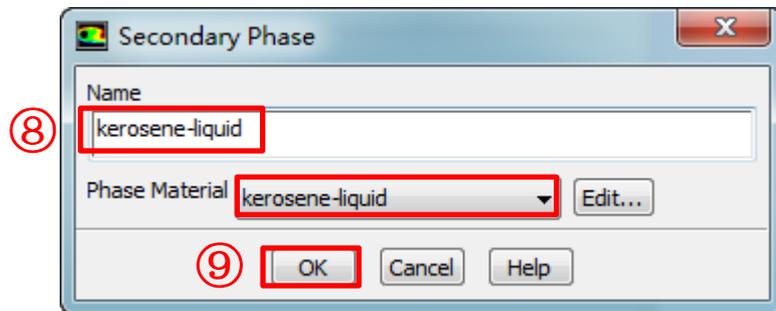
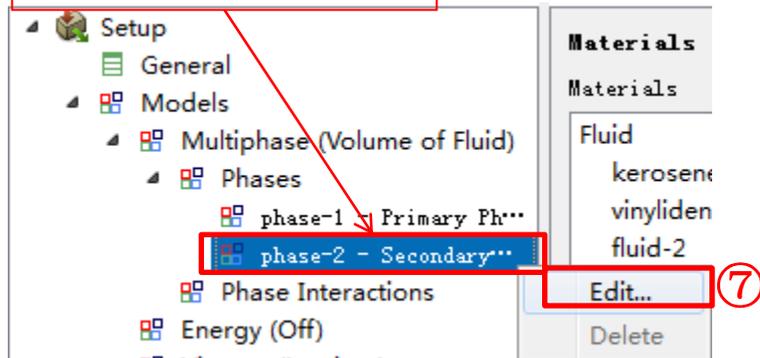
物理模型设定

气液相设定

右键点击Phase-1-Primary Phase



⑥ 右键点击，点击edit



计算设定

物理模型设定

求解参考值设定

① Boundary Conditions

② Operating Conditions...

③ Operating Pressure (pascal)

④ Reference Pressure Location

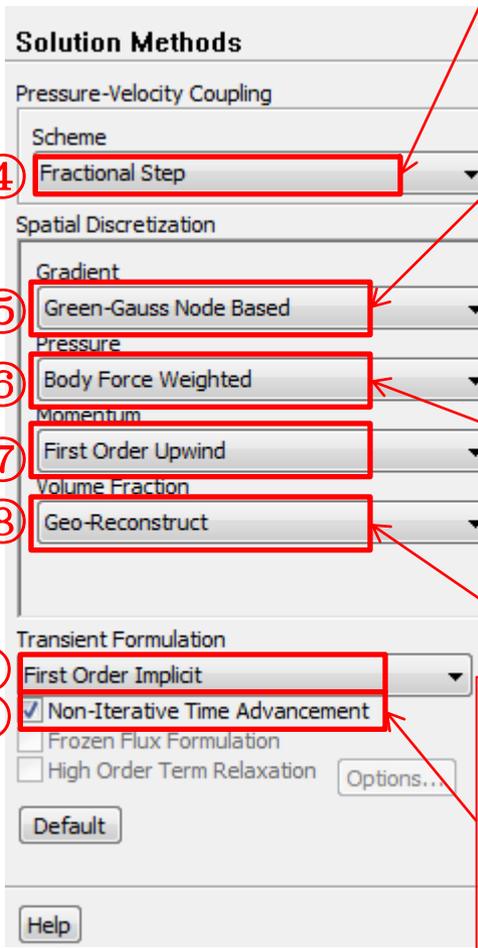
⑤ OK

参考压力取值位置, 一般设定为 Primary phase区域内, 本例设定空气相中

重力加速及油箱内流体运动加速方向, 详见P9设定

计算设定

求解器设定



The screenshot shows the 'Solution Methods' panel in ANSYS Fluent. The left sidebar has 'Solution Methods' highlighted with a red box and a circled '1'. The main panel has several dropdown menus highlighted with red boxes and circled numbers 2 through 8. Red arrows point from text boxes to these settings.

① Solution Methods

② Non-Iterative Time Advancement

③ First Order Implicit

④ Fractional Step

⑤ Green-Gauss Node Based

⑥ Body Force Weighted

⑦ First Order Upwind

⑧ Geo-Reconstruct

① 一阶隐式方程，一般可以应对绝大多数求解，要求更高，可以改为Couple

Fractional Step求解精度和稳定性不如PISO好，但是求解速度快，调整Pressure Release为0.7~0.8甚至更小

Green-Gause Cell based可能出现伪扩散；**Green-Gause Node based**最小化的伪扩散推荐用于三角形的网格上；**Least squares cell based**基于单元体的最小二乘法差值，推荐用于多面体网格上与**Green-Gause Node based**精度相同

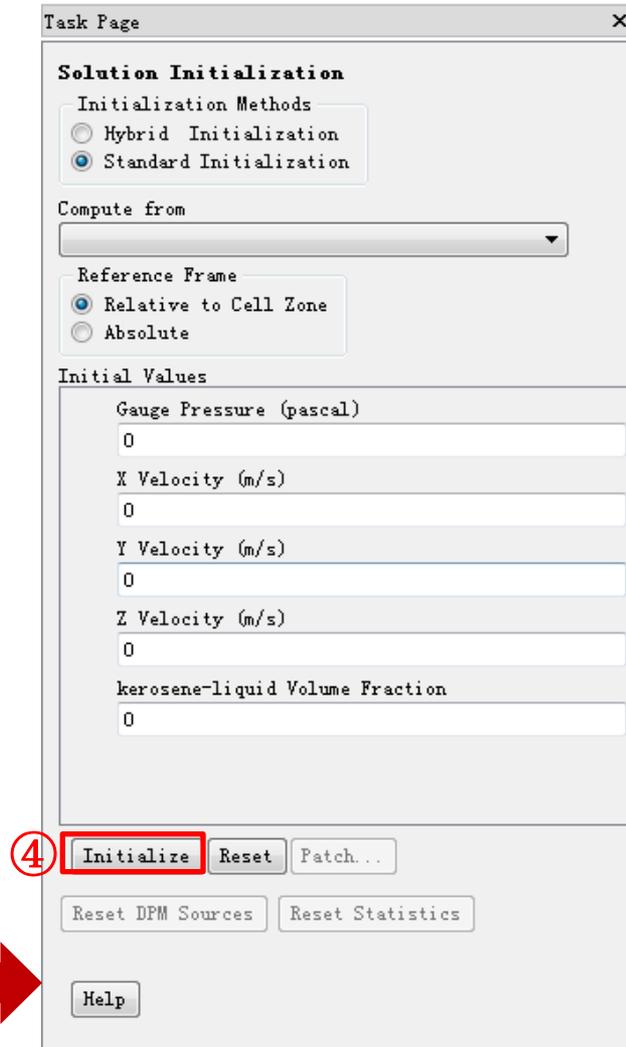
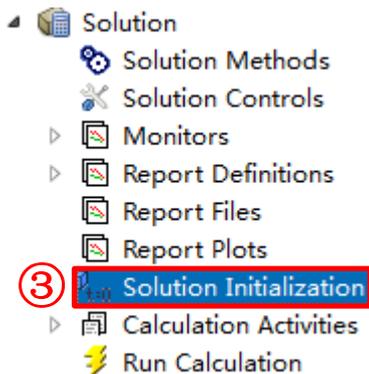
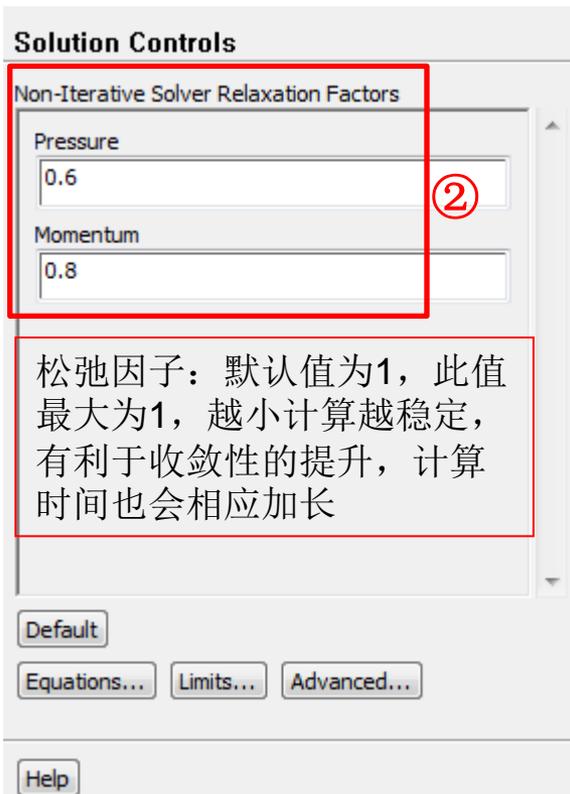
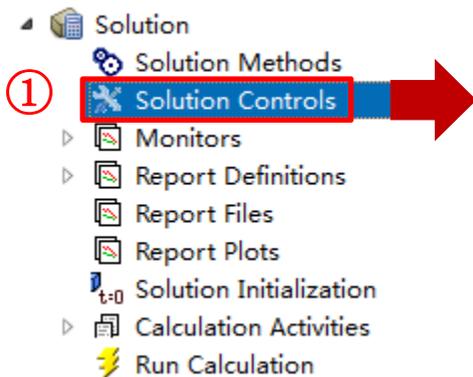
PRESTO: 适用于高雷诺数自然对流、强涡流、高速转动和转弯弧度很大的计算域；
Body Force Weighted: 适用于体积力较大的问题，如油箱晃动

基于几何信息追踪交界面的一种算法，精度最高，但硬件资源消耗最大，当求解问题网格质量不高时推荐使用

非迭代时间推进法。是专门针对非稳态问题的一种算法，一般与**PISO**算法联合使用，称为瞬态**PISO**算法。与稳态问题的区别，在瞬态计算的每个时间步内，利用**PISO**算法计算时，不需要迭代，**PISO**算法的精度取决于时间步长，使用越小的时间步长，可取得越高的计算精度，当时间步长比较小时，不进行迭代，也可以保证计算有足够的精度

计算设定

求解器设定



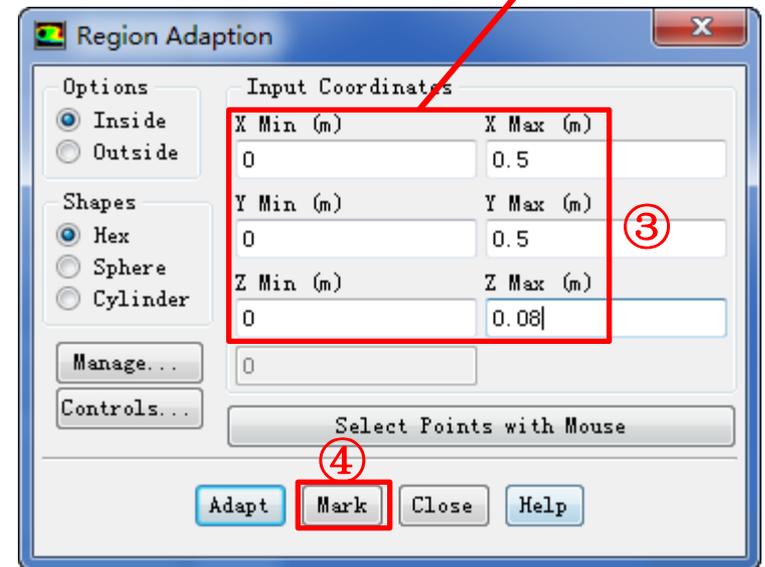
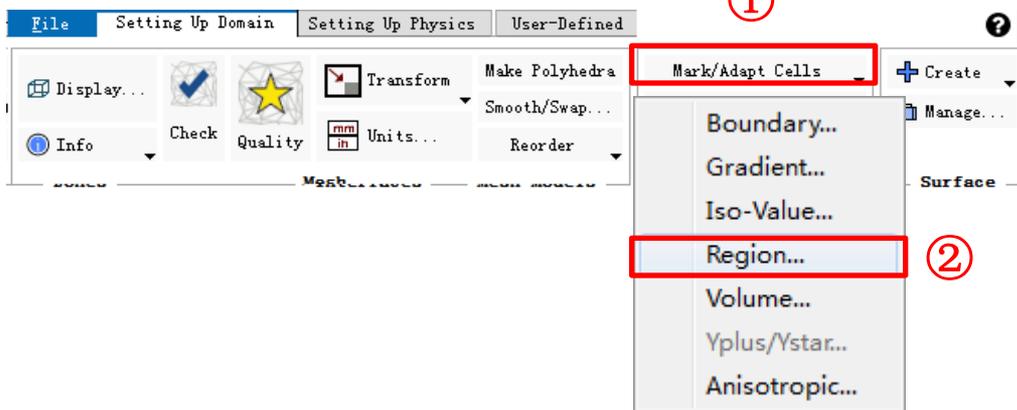
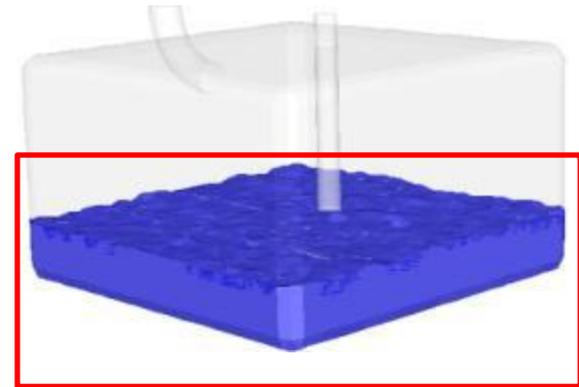
计算设定

■ 求解器设定

◆ 标记需要初始化液体区域

➢ 标记区域可以用于液相（油）的初始化

注意：Adapt进行网格细化操作，此处不需要点击此按钮

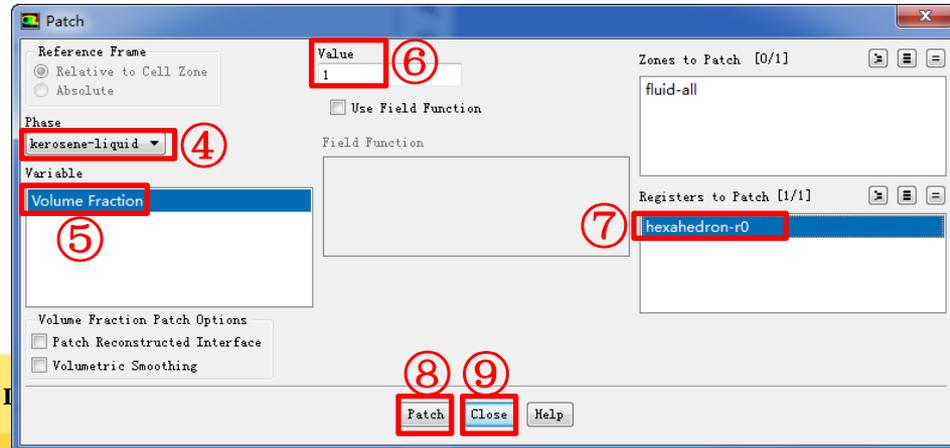
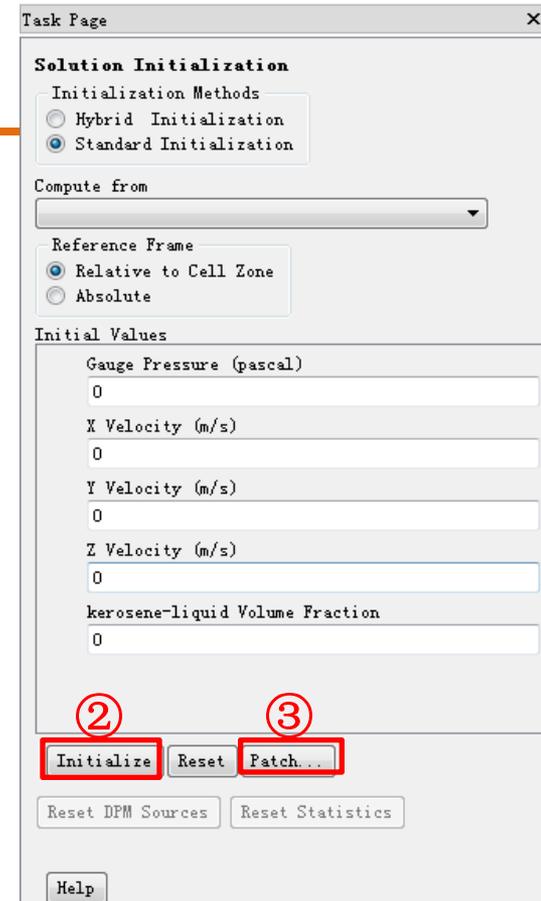
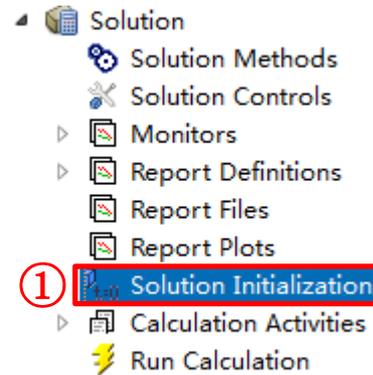


计算设定

■ 求解器设定

◆ 初始化液面区域

- 1) 对P16页Mark的hexahedron-r0区域进行初始化
- 2) 初始化区域内为Kerosene-liquid (油相) 体积分数为值1



计算设定

后处理

监控

①

②

③

④

⑤

左键点击

Monitors

Residuals, Statistic and Force Monitors

Residuals - Print, Plot

Statistic - Off

Create Edit... Delete

Surface Monitors

Residual Monitors

Options

Print to Console

Plot

Window: 1

Iterations to Plot: 250

Iterations to Store: 1000

Equations

Residual	Monitor	Check Convergence	Absolute Criteria
continuity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.001
x-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.001
y-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.001
z-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.001

Residual Values

Normalize

Scale

Compute Local Scale

Iterations: 5

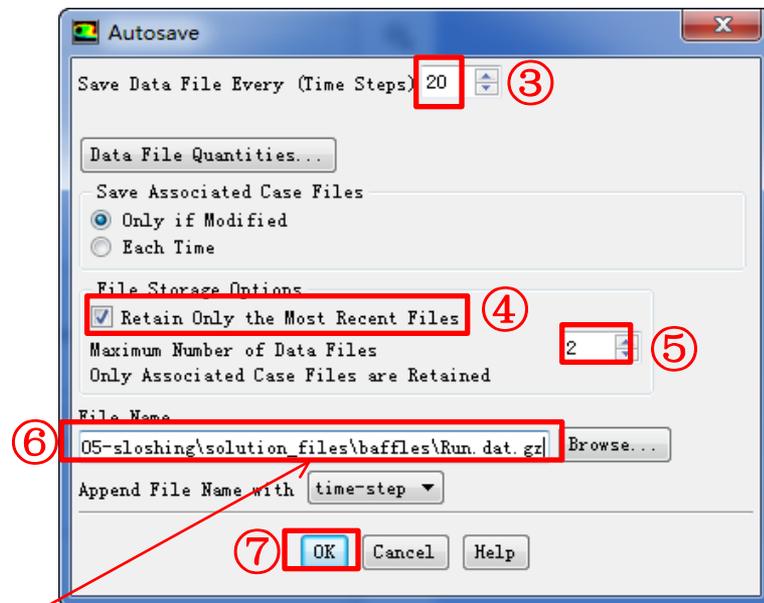
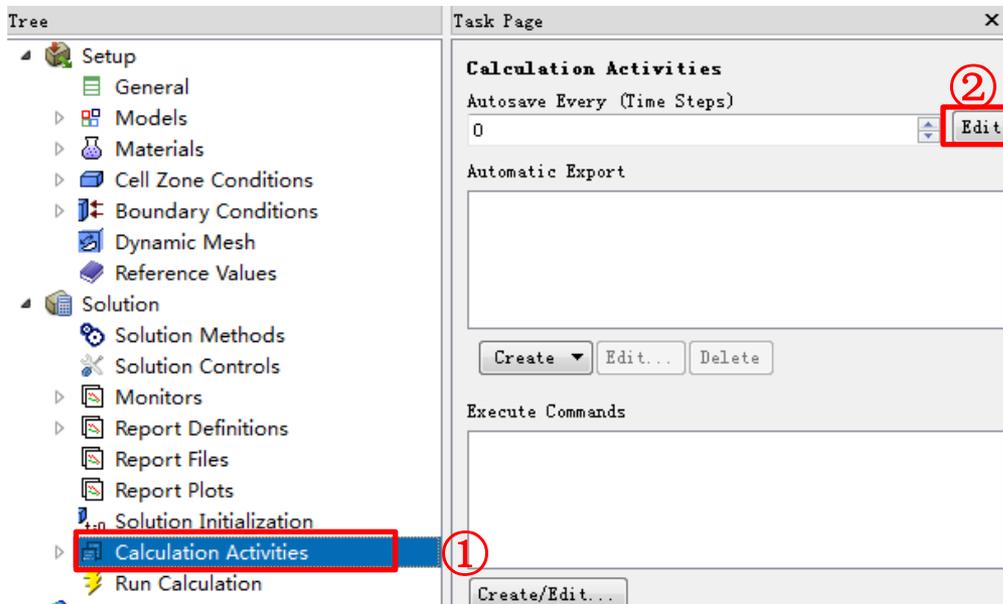
OK Plot Renormalize Cancel Help

计算设定

■ 后处理设定

◆ 自动存储计算结果文件

- 1) Save Data File Every (Time Steps) : 20个时间步存储一次结果文件
- 2) Retain Olly the Most Recent Files: 覆盖式存储, 设定保留文件个数



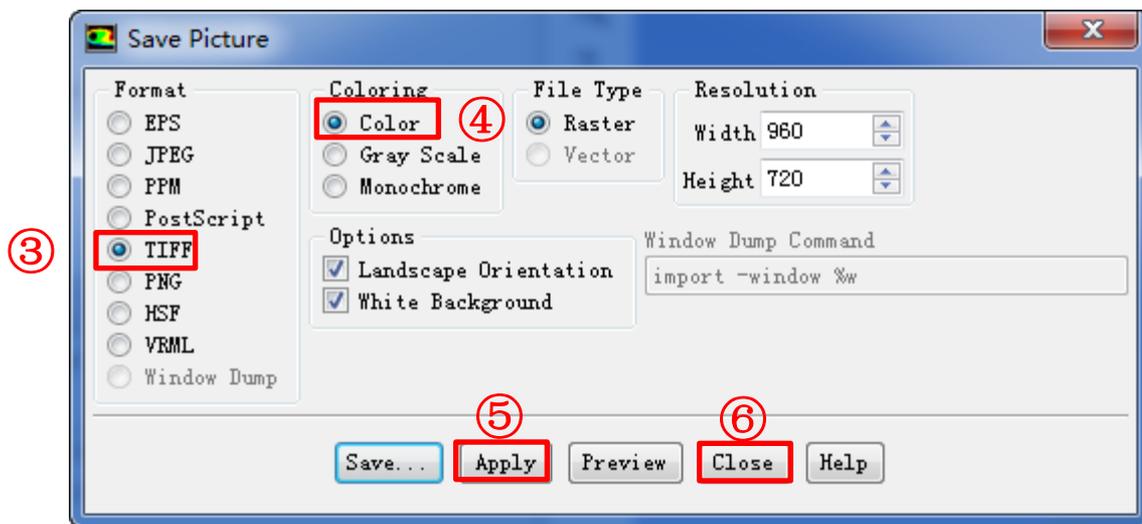
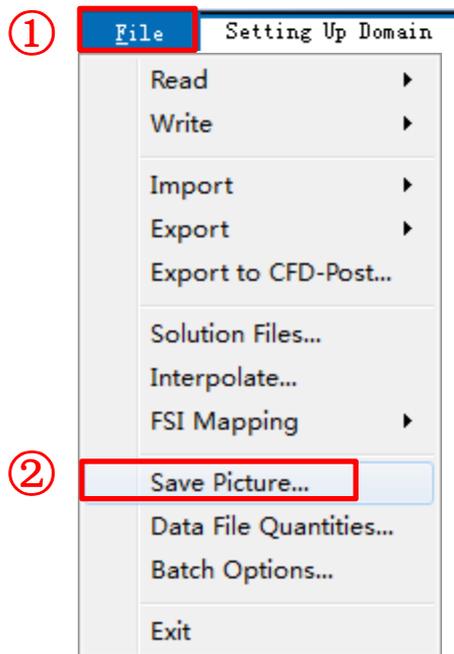
存储工程文件名称, 文件后缀.dat.gz

计算设定

■ 后处理设定

◆ 动画设定

➤ 存储图片格式

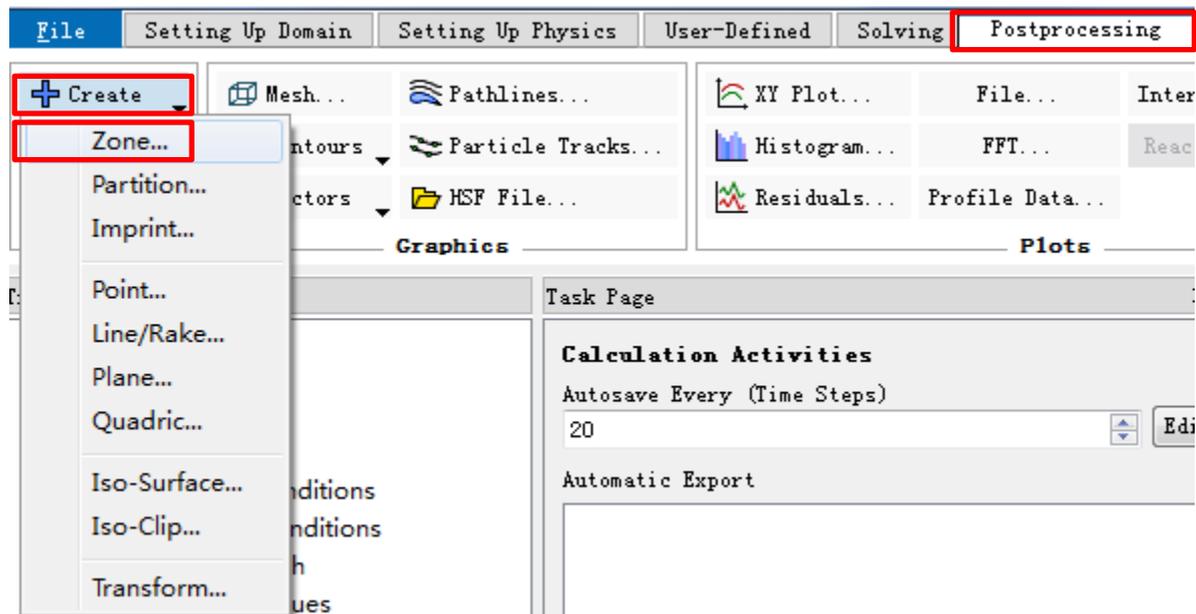
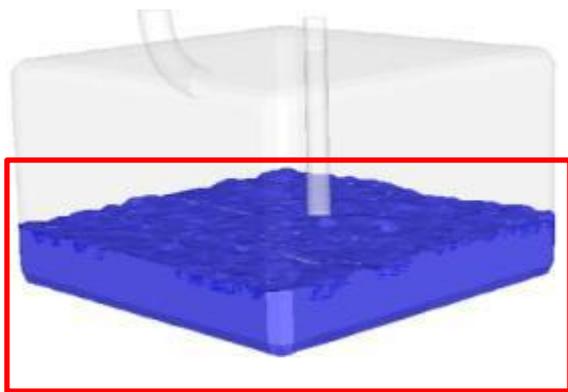


计算设定

①

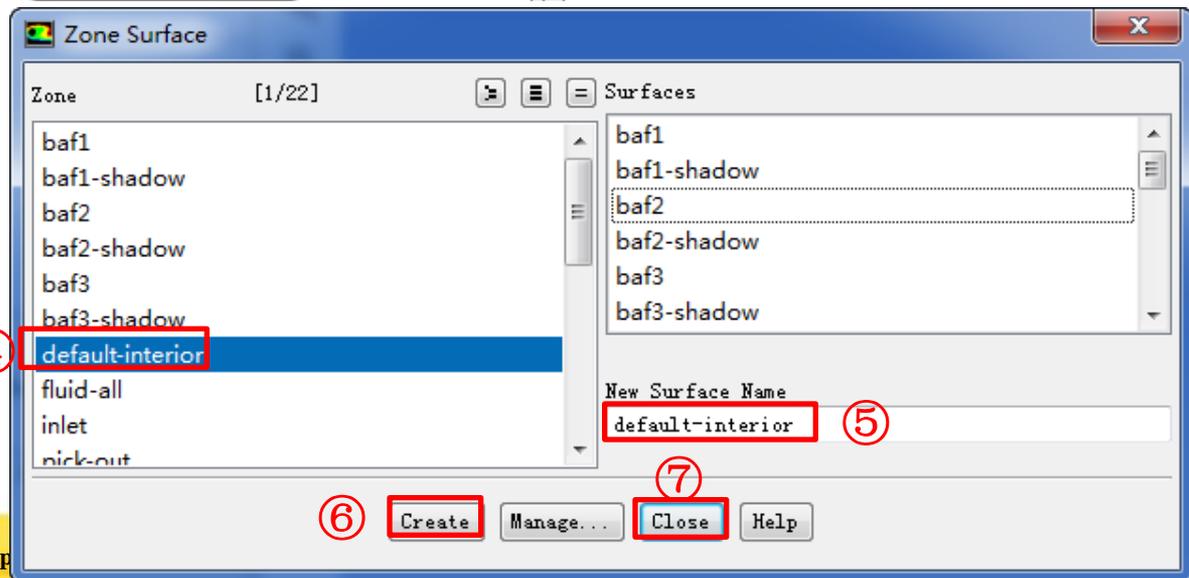
后处理设定

- ◆ 创建一个基于流体区域的表面



②

③



④

⑥

⑦

⑤

计算设定

■ 后处理设定

◆ 活动窗口设定

点击上下箭头，调整活动窗口序号，Open打开口2，Set将窗口2激活为当前活动窗口

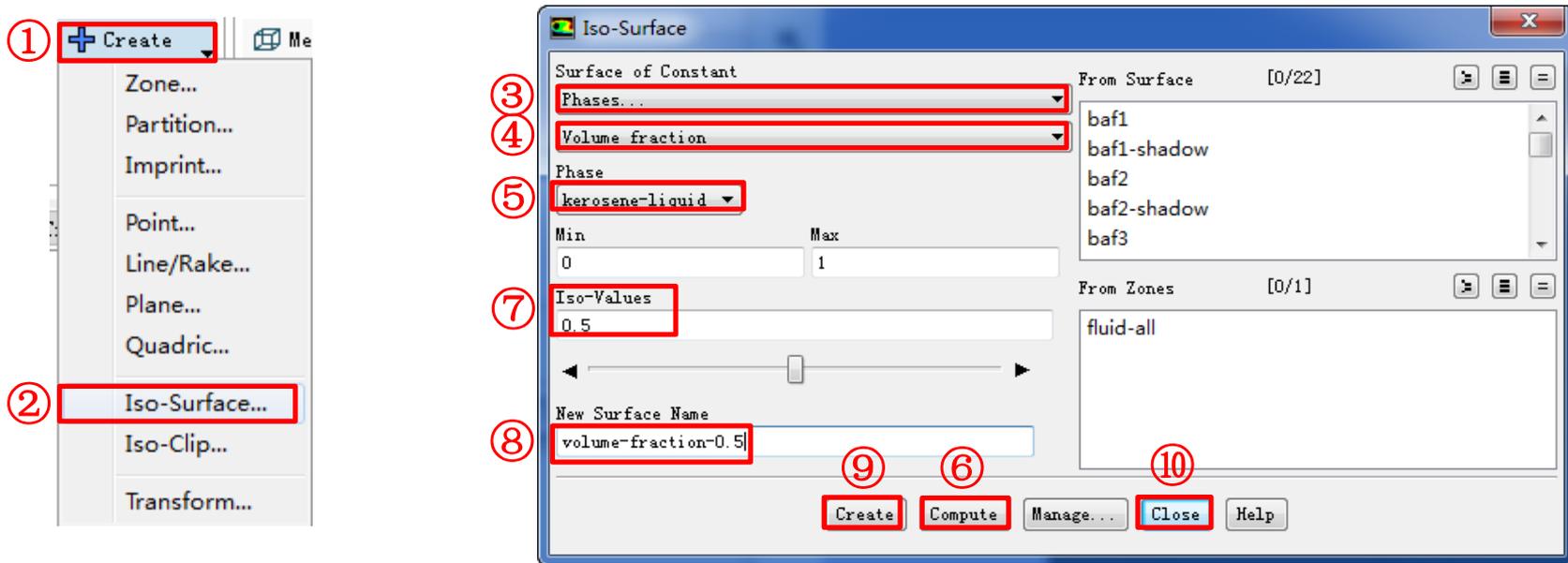
The screenshot displays the ANSYS Fluent software interface. On the left, the 'Tree' panel shows the 'Animations' folder selected (1). The 'Task Page' panel shows the 'Graphics and Animations' task page with 'Mesh' selected under 'Graphics' and 'Sweep Surface' selected under 'Animations'. The 'Options...' button is highlighted (2). The 'Display Options' dialog box is open, showing the 'Rendering' and 'Graphics Window' sections. The 'Active Window' dropdown is set to '2' (3), and the 'Open' button is highlighted (4). The 'Set' button is highlighted (5). The 'Lighting' dropdown is set to 'Phong' (6). The 'Apply' button is highlighted (7), and the 'Close' button is highlighted (8).

计算设定

■ 后处理设定

◆ 创建等值面

- 通过**Compute**获取流体域域中油相体积分数范围
- 油相体积分数**0.5**，作为油气两相液面的分界线

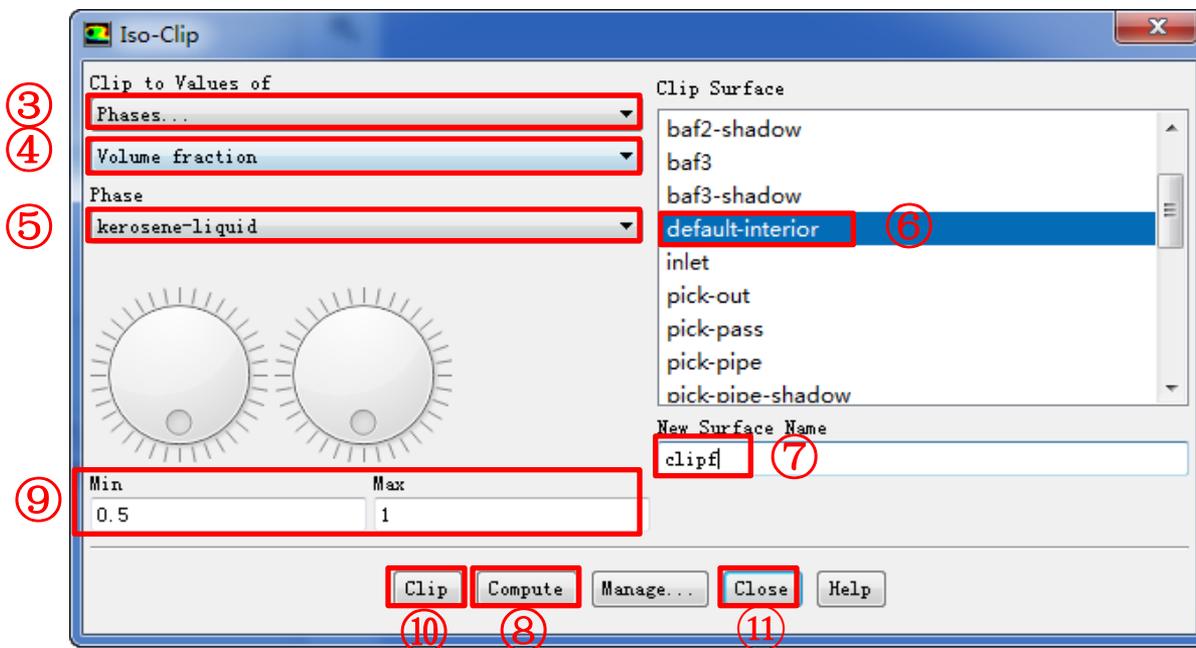
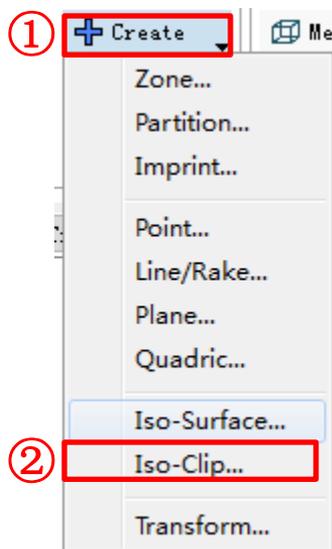


计算设定

■ 后处理设定

◆ 创建等值油相表面

- 油相体积分数为0.5~1之间的网格外表面创建一个面

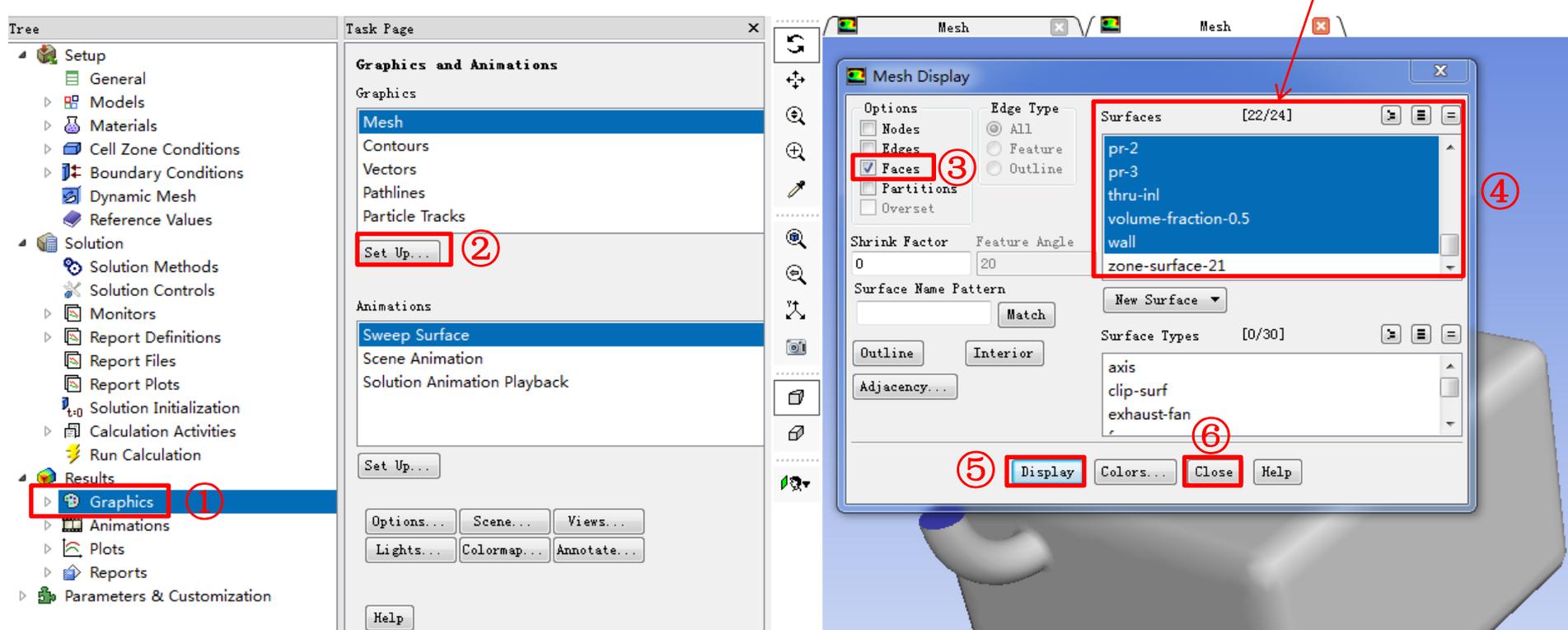


计算设定

后处理设定

显示面

选择除zone-surface-21和default interior意外所有的面



计算设定

■ 后处理设定

◆ 调色

➤ 油相颜色调整

选择clipf (体等值面)
和volume-fraction-0.5
(等值面)

The screenshot shows the ANSYS Fluent software interface. On the left, the 'Tree' panel has 'Graphics' highlighted with a red box and the number 1. In the center, the 'Task Page' has 'Scene...' highlighted with a red box and the number 2. The 'Scene Description' dialog box is open, with 'volume-fraction-0.5' selected in the 'Names' list, highlighted with a red box and the number 3. The 'Display...' button in the 'Geometry Attributes' section is highlighted with a red box and the number 4. On the right, the 'Display Properties' dialog box is open, with the 'Colors' section highlighted with a red box and the number 6. The 'Apply' button in the 'Display Properties' dialog is highlighted with a red box and the number 7, and the 'Close' button is highlighted with a red box and the number 8.

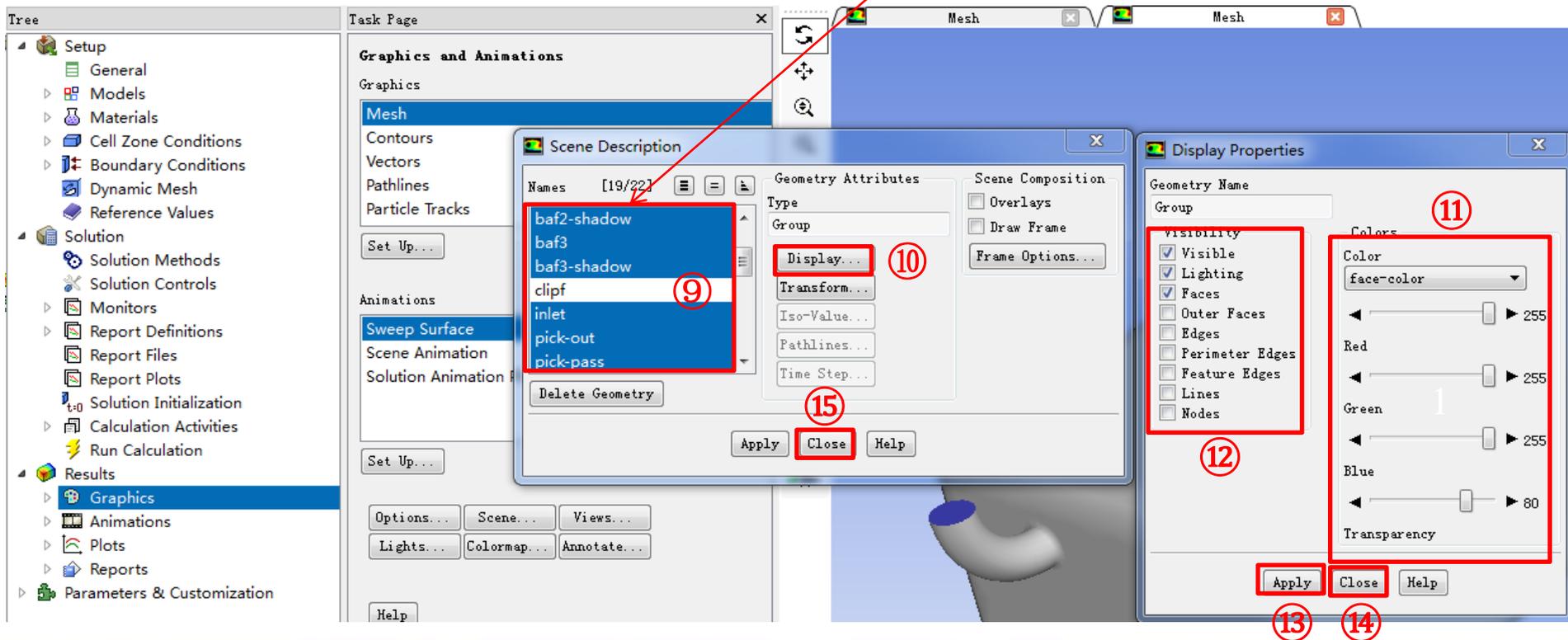
计算设定

■ 后处理设定

◆ 调色

➢ 油箱外表面颜色调整

选择除clipf和volume-fraction-0.5意外的所有表面

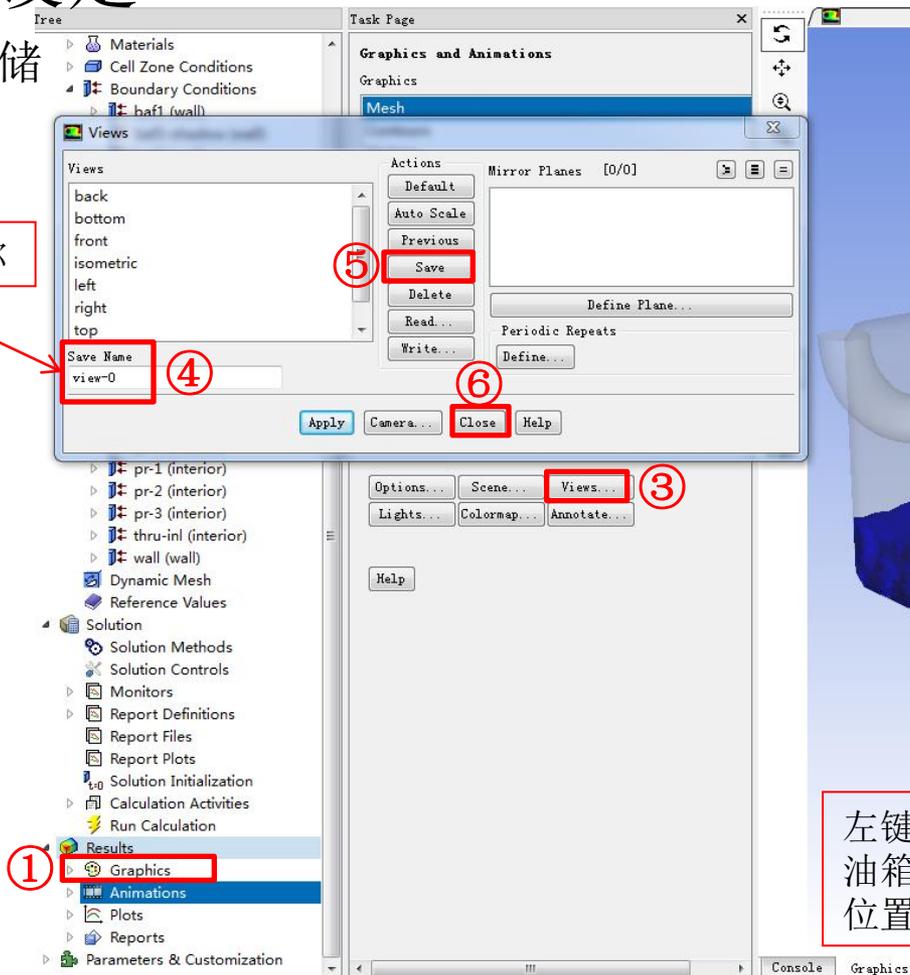


计算设定

后处理设定

视角存储

自定义视角名称



左键点击视窗空白处旋转油箱角度至方便查看液面位置的角度

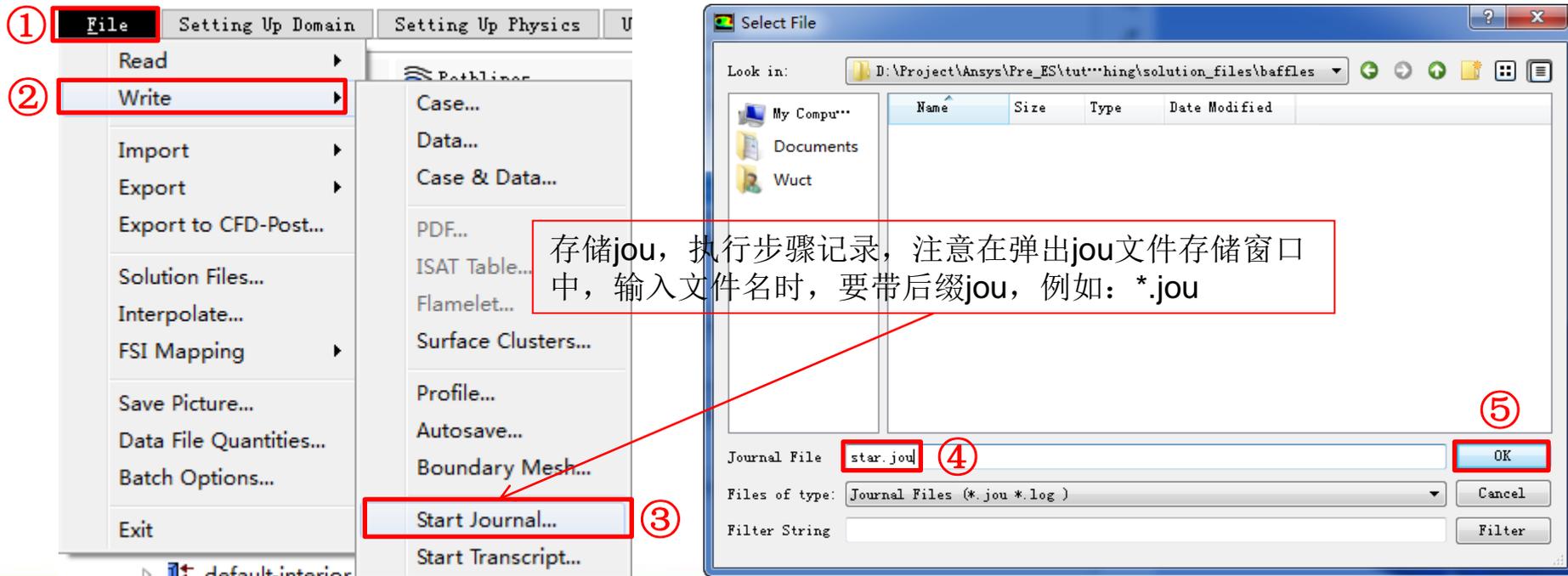
②

计算设定

■ 后处理设定

◆ 录制设定

- 1) Start Journal: 开始录制
- 2) Stop Journal: 停止录制, 保存录制文件
- 3) 调用*.jou文件, 执行录制过程

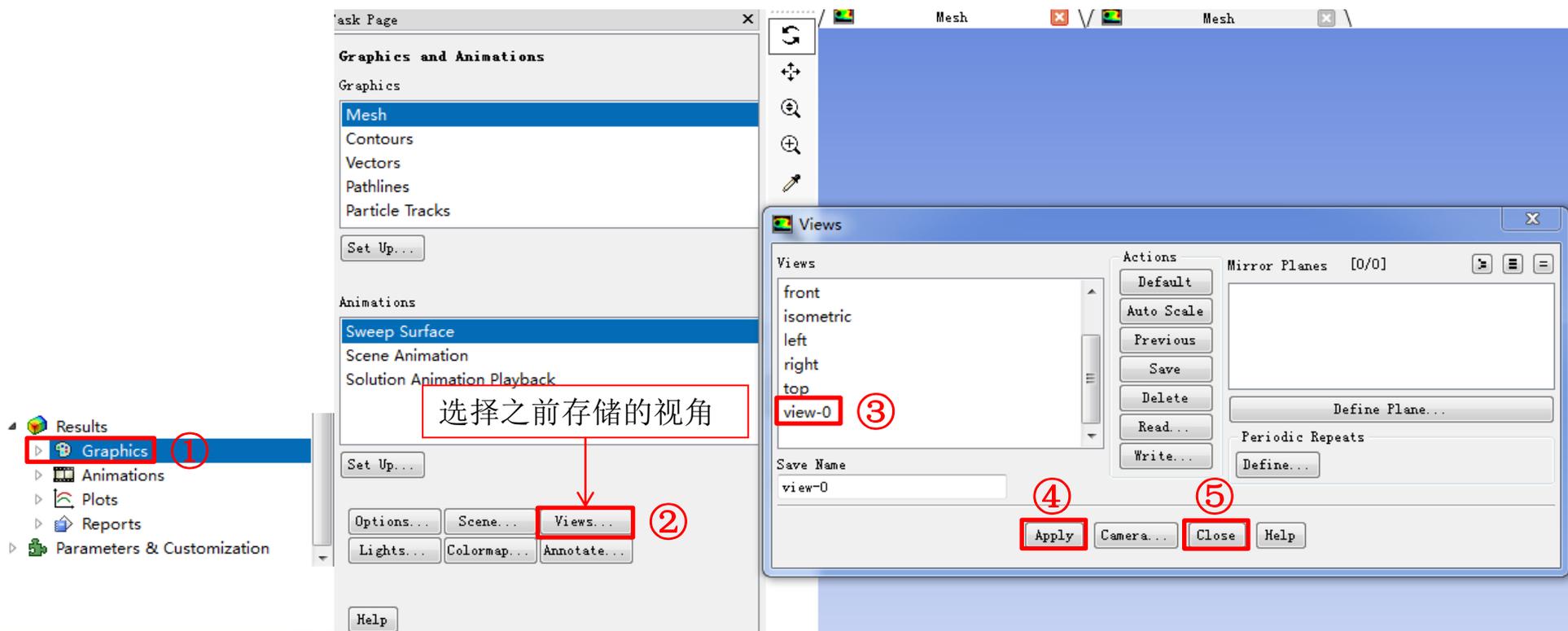


计算设定

■ 后处理设定

◆ 录制设定

➤ 视角调整

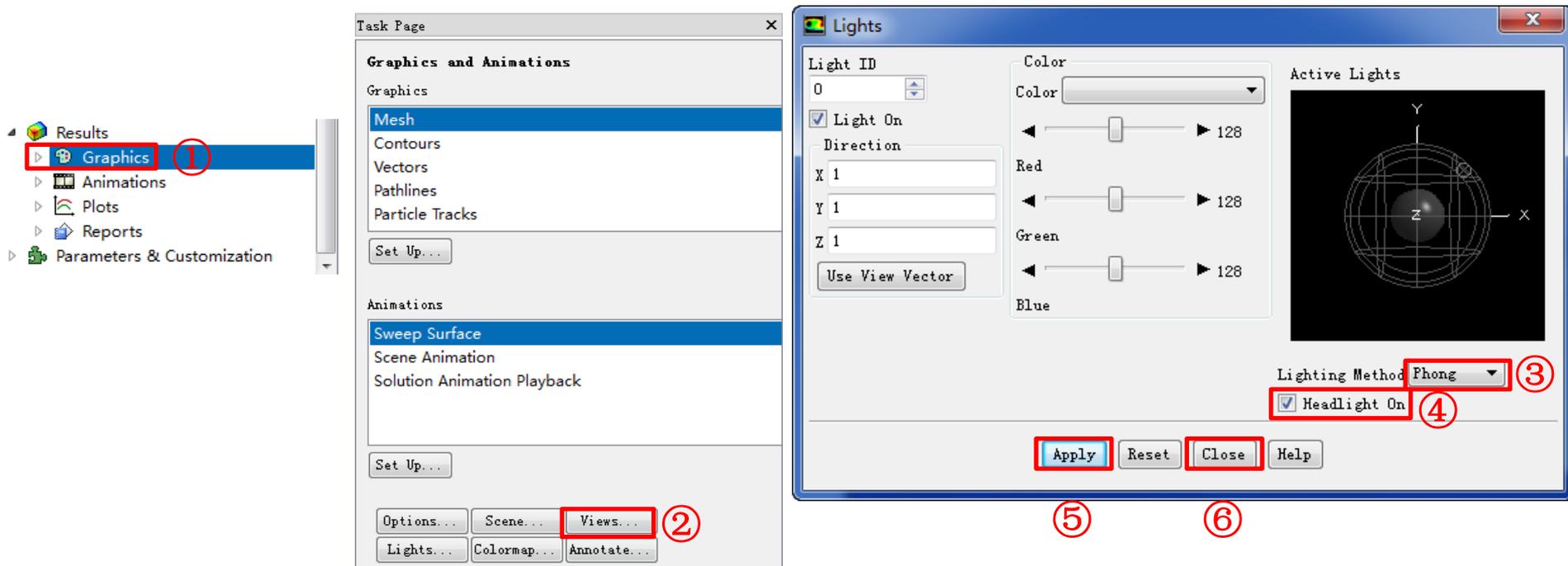


计算设定

■ 后处理设定

◆ 录制设定

➤ 灯光调整

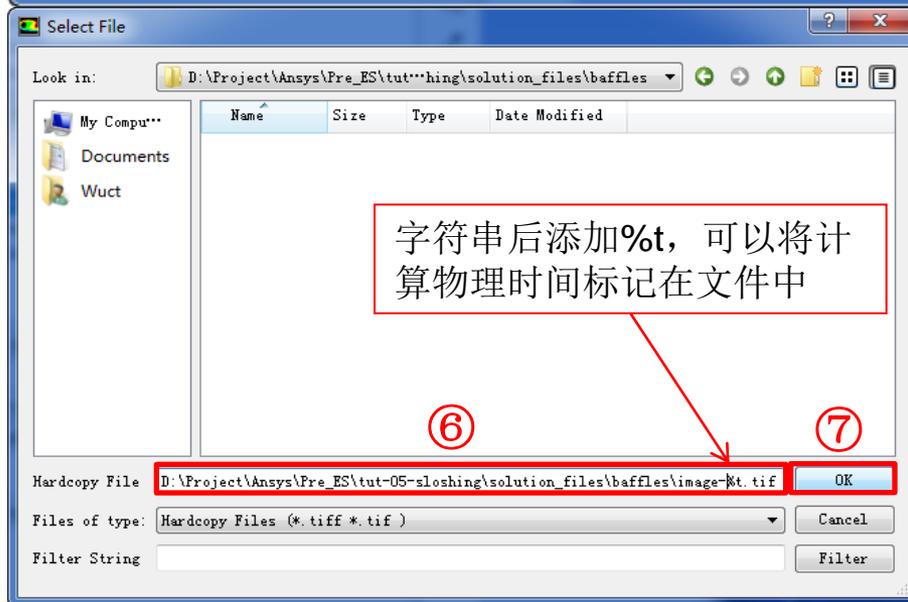
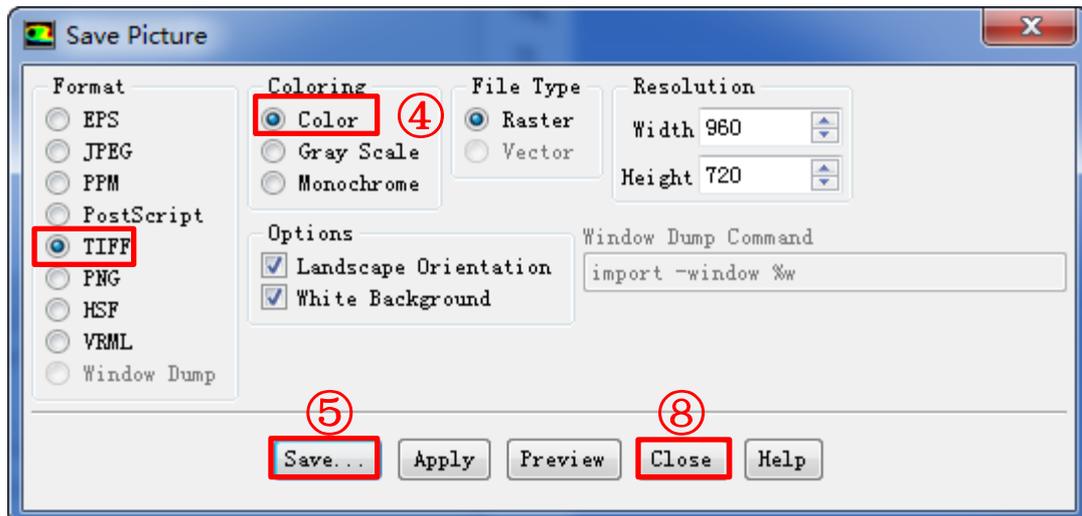
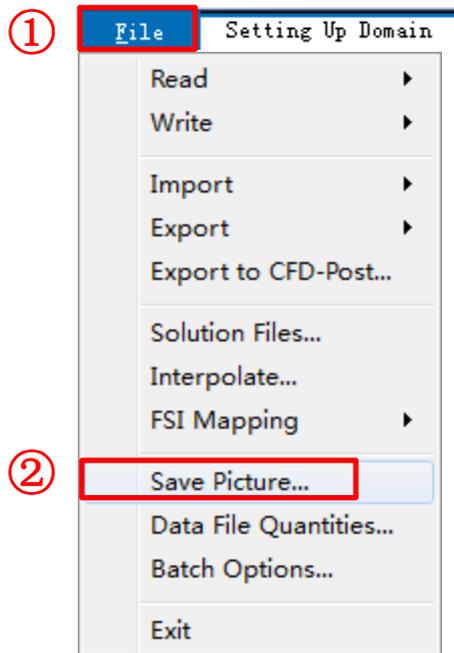


计算设定

■ 后处理设定

◆ 录制设定

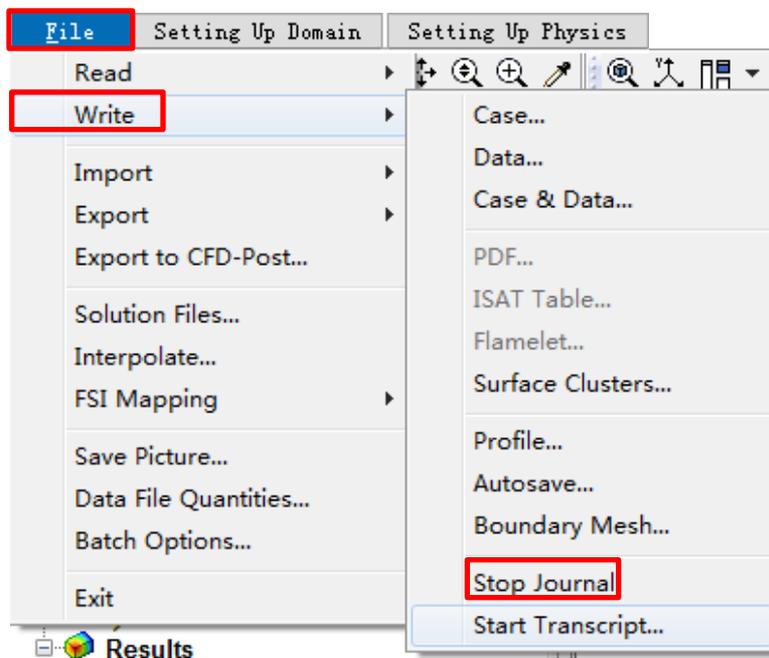
➤ 随时间步存储图片



计算设定

■ 后处理设定

◆ Stop Journal: 结束设置录制



计算设定

■ 后处理设定

◆ 通过执行命令循环截取图片

The screenshot displays the IDA software interface. On the left is the 'Tree' panel with 'Calculation Activities' selected. The 'Task Page' shows 'Calculation Activities' with 'Autosave Every (Time Steps)' set to 100. The 'Execute Commands' section lists 'command-1 - Active', 'command-2 - Active', and 'command-3 - Active'. A 'Create/Edit...' button is highlighted with a red circle 2. An 'Execute Commands' dialog box is open, showing a table of defined commands. The table has columns for 'Active Name', 'Every', 'When', and 'Command'. Three commands are listed: 'command-1' (Every: 20, When: Time Step, Command: display set-window 2), 'command-2' (Every: 20, When: Time Step, Command: file read-journal start.jou), and 'command-3' (Every: 20, When: Time Step, Command: display set-window 1). The 'Defined Commands' dropdown is set to 3 (circle 3). The 'OK' button is highlighted with a red circle 5. A blue text box on the right contains the following text:

三条自动执行命令：
1. display set-window 2, 将窗口2设定为活动窗口；
2. file read-journal start.jou 执行截取图片操作；
3. display set-window 1, 将窗口1设定为活动窗口。
以上3条命令每隔20个时间步自动顺序执行一遍。

1

2

3

4

5

计算设定

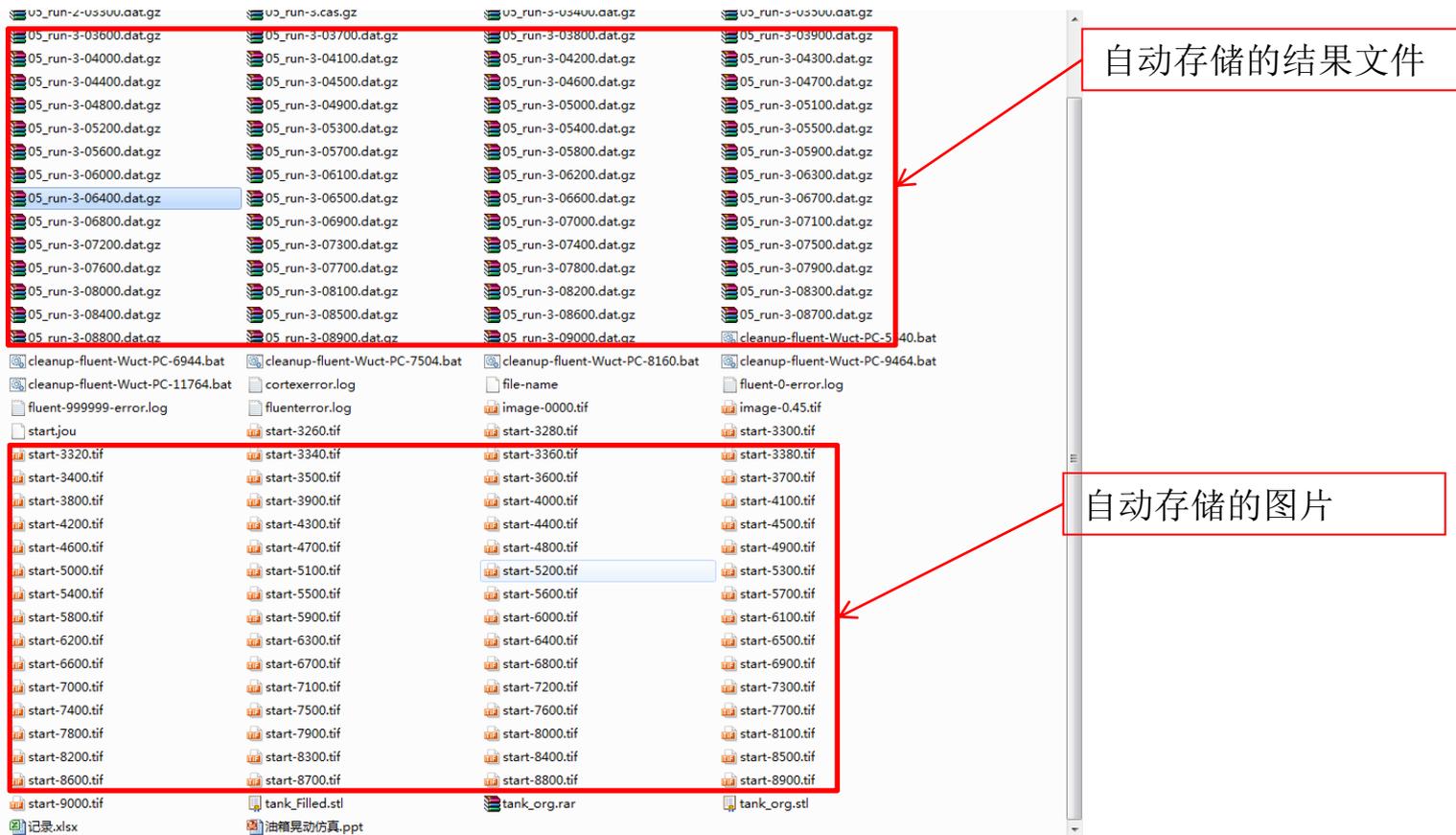
■ 执行计算

- 1) Variable Time Step: 自适应可变时间步长
- 2) Time Step Size (s): 时间步长
- 3) Number of Time Steps: 总迭代步数

The screenshot displays the software's calculation setup interface. On the left, the 'Tree' panel shows the 'Run Calculation' option selected. The main 'Task Page' is titled 'Run Calculation' and contains several fields: 'Time Stepping Method' is set to 'Variable', 'Time Step Size (s)' is 0.0001, and 'Number of Time Steps' is 10000. Below these are 'Options' for extrapolation and data sampling, and 'Reporting Interval' and 'Profile Update Interval' both set to 1. A 'Calculate' button is at the bottom. A 'Variable Time Step Settings' dialog box is overlaid on the right, with fields for 'Global Courant Number' (2), 'Ending Time (s)' (1.25), 'Minimum Time Step Size (s)' (1e-05), 'Maximum Time Step Size (s)' (0.0025), 'Minimum Step Change Factor' (0.5), 'Maximum Step Change Factor' (1.5), 'Number of Fixed Time Steps' (1), and 'User-Defined Time Step' (none). The 'OK' button in the dialog is highlighted.

计算结果输出

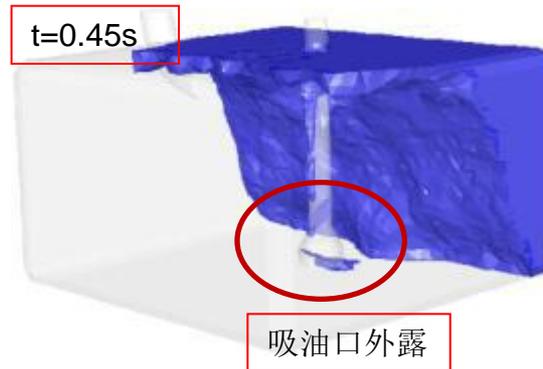
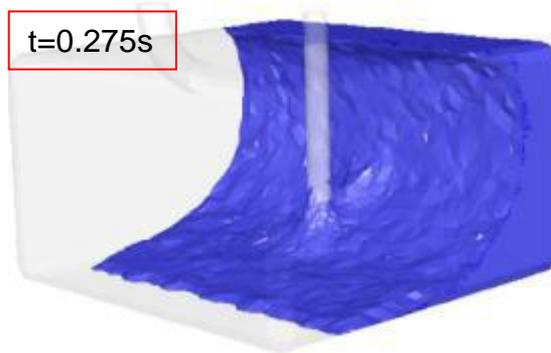
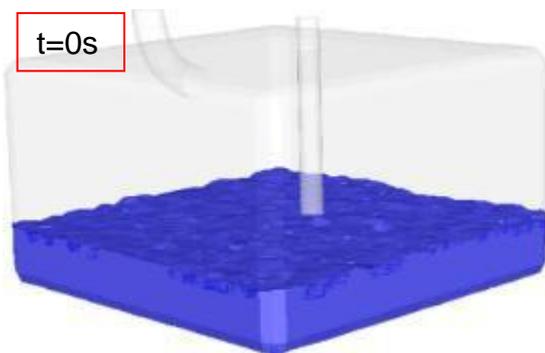
■ 计算输出



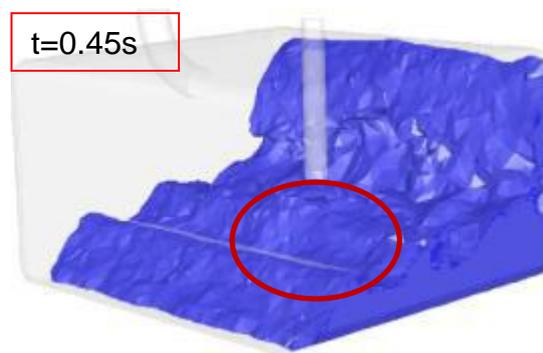
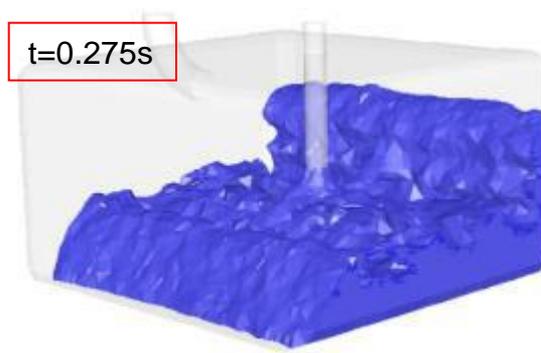
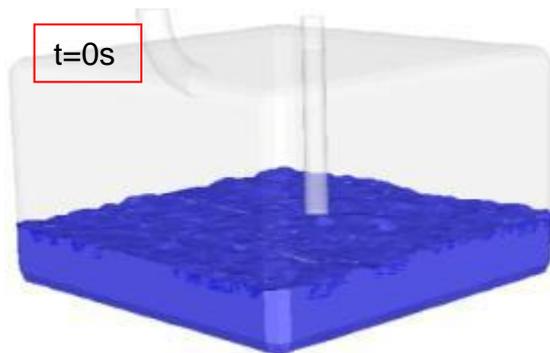
计算结果输出

■ 计算输出

无防浪板



有防浪板



注：结果引用自《CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing》

目录

■ 概述

■ 模型简介

■ 计算设定

- ◆ 软件启动

- ◆ 网格处理

- ◆ 物理模型设定

- ◆ 求解器设定

- ◆ 后处理设定

 - 计算输出设定

 - 瞬态计算图片输出

- ◆ 提交计算

■ 总结

总结

- ANSYS Fluent具有模型全面，设定界面清晰简单的特点
- VOF模型，具有快速准确的特点，方便实现燃油箱晃动、注油、齿轮箱搅油等物理现象
- ANSYS Fluent计算结果输出全面，方便后期分析之用
- IDAJ公司提供基于VOF模型的燃油箱晃动、注油、齿轮箱搅油等最佳实践方案

关注微信公众号“IDA J-China”
获得及时的信息/培训/活动资讯



谢谢大家!

艾迪捷信息科技有限公司（北京）
北京市朝阳区光华路甲14号1幢
诺安基金大厦1601室

Email: support@idaj.cn

wu.chengtao@idaj.cn

电话: (010) 65881497/65881498

传真: (010) 65881499

邮编: 100020