

CAELinux Tutorial

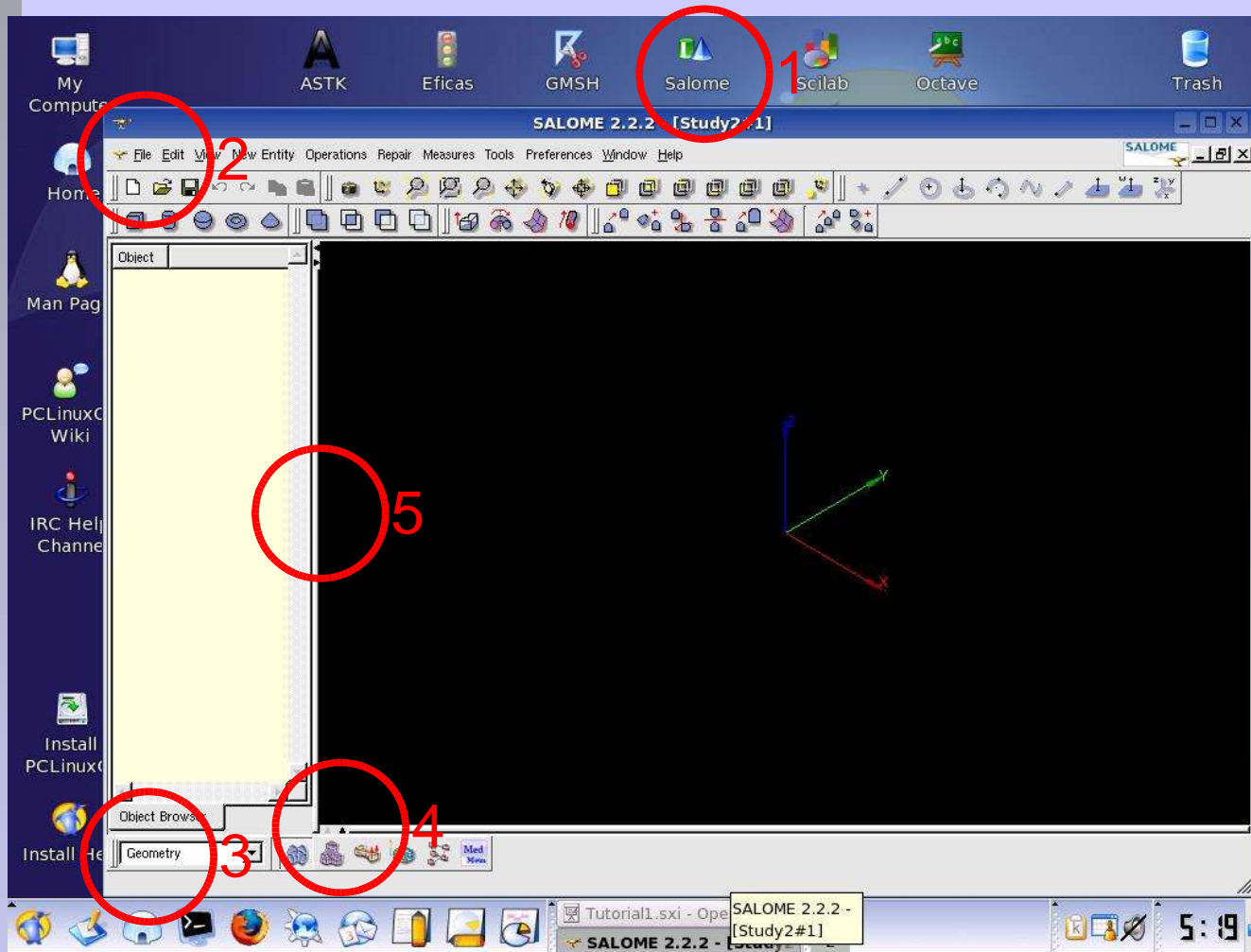
**Finite element analysis
with CAELinux:**

**Tutorial 1
3D Geometry Modelling & Meshing
in SALOME**

Overview

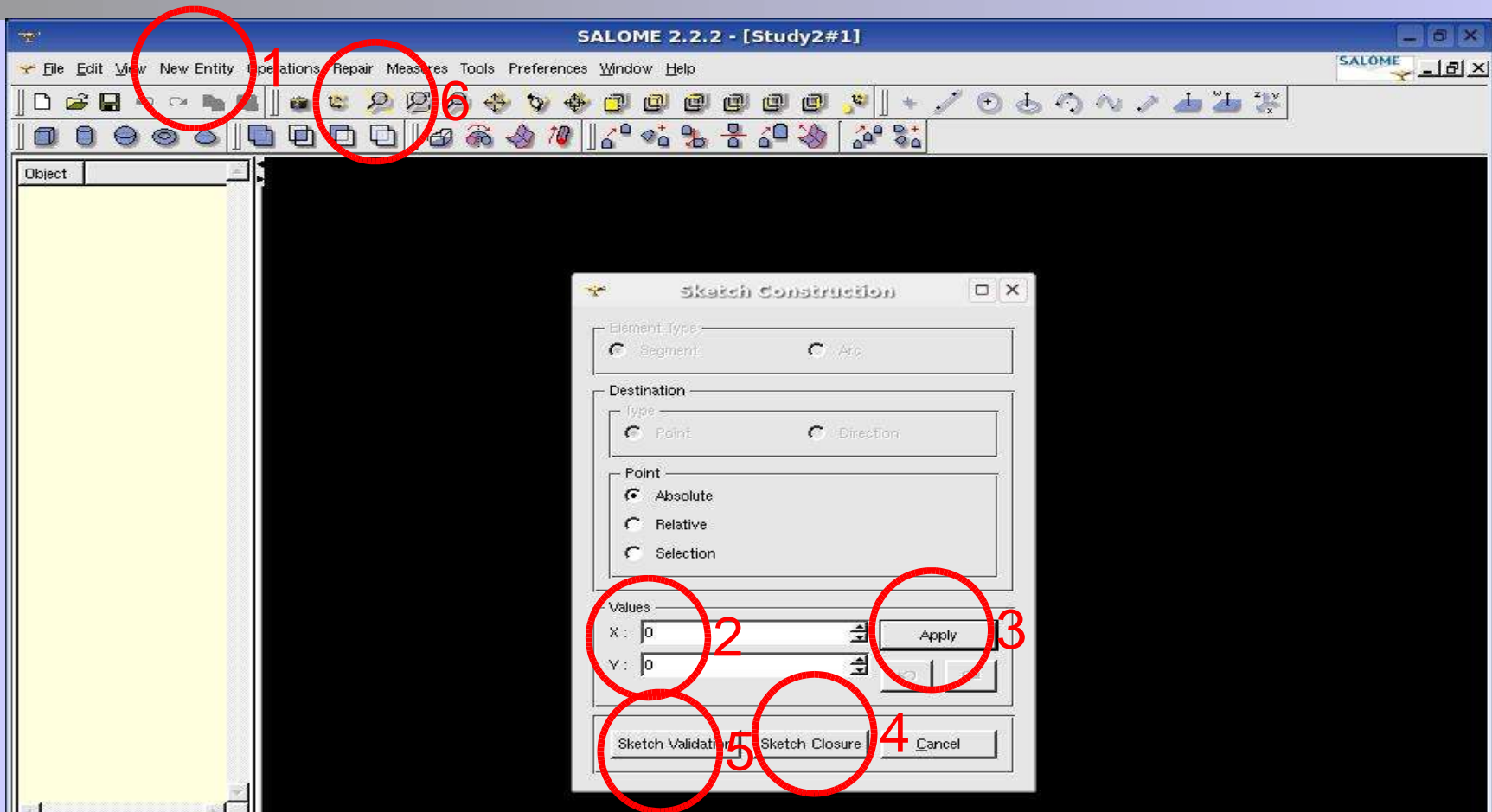
- ◆ Geometry & mesh (Salome)
 - ◆ Geometry modelling (Salome/Geometry)
 - ◆ Meshing & group definition (Salome/Meshing)
- ◆ Finite Element Analysis (Code-Aster)
 - ◆ Create a FE study profile (ASTK)
 - ◆ Definition of the FEA (Eficas)
 - ◆ Running the analysis (ASTK)
 - ◆ Reading the solver's messages (ASTK)
- ◆ Post processing (Salome)
 - ◆ Importing results from MED file
 - ◆ Displacements / strains / stresses

Start Salome / Geometry



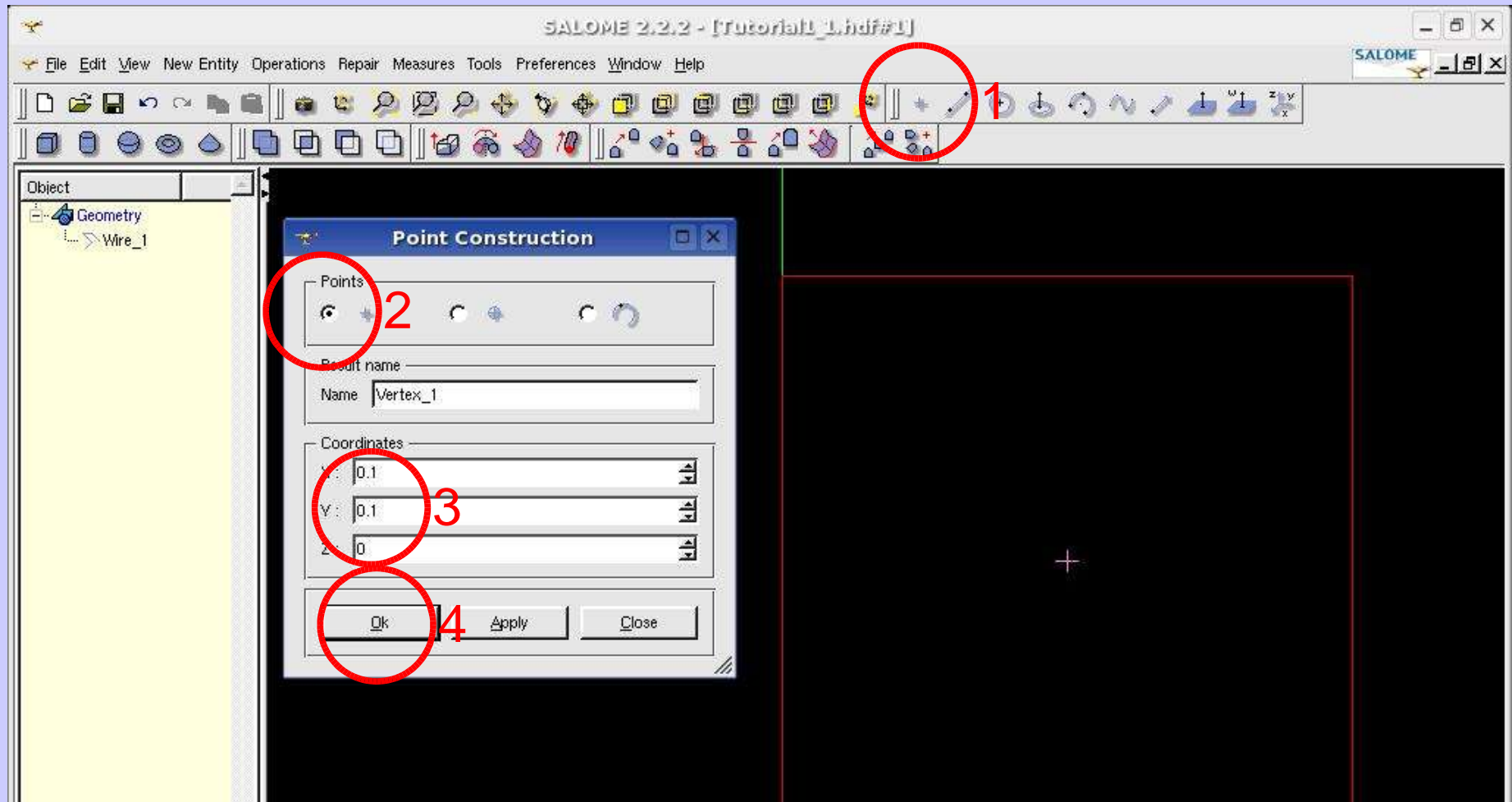
1. start Salome
2. File-> New
3. Select Geometry
4. Reduce Console view (small arrow)
5. Resize Tree view

Create Geometry: 2D Sketch



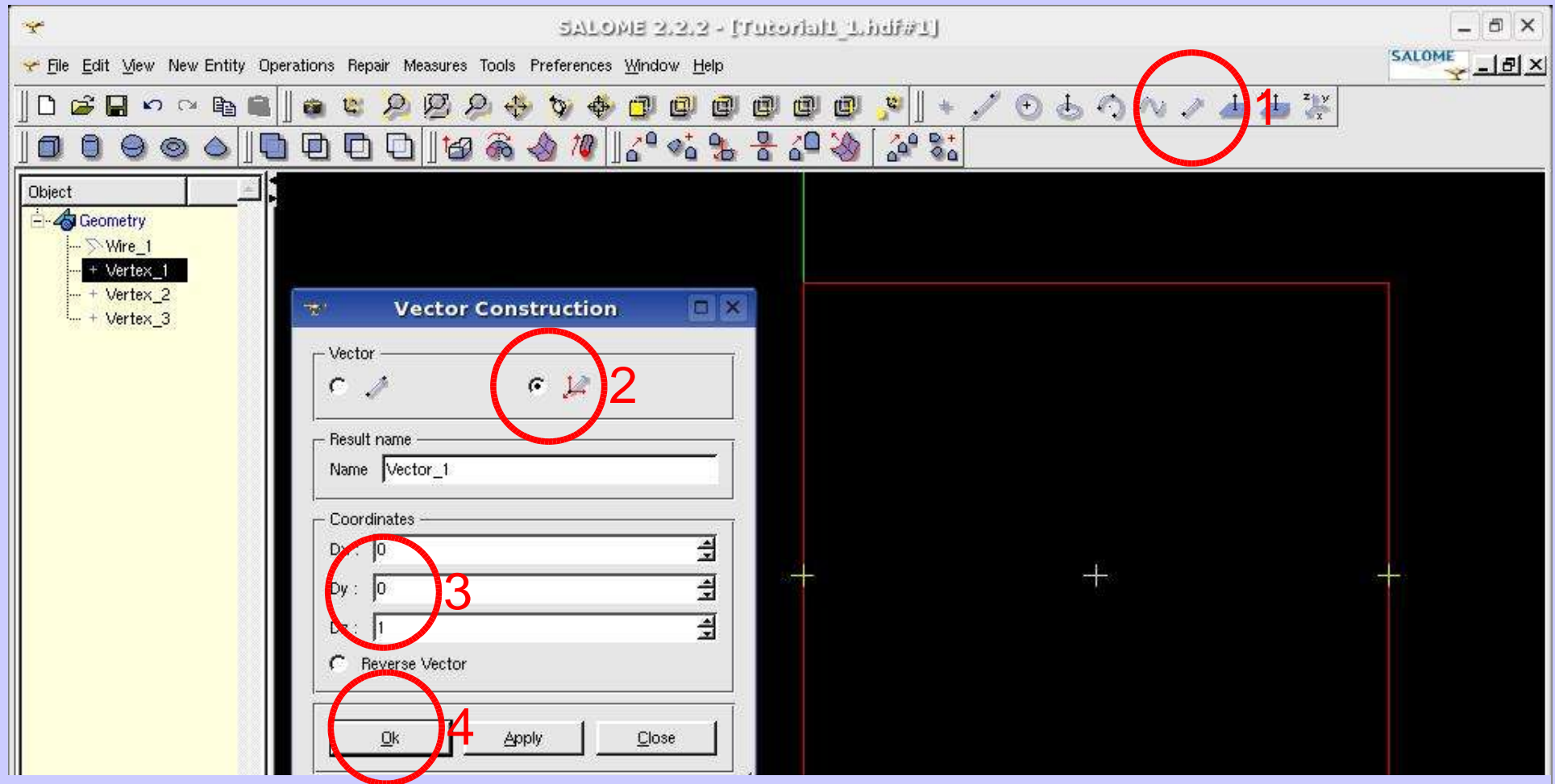
1. New Entity -> Sketch (1)
2. In Sketch Construction (2): enter these coordinates (0;0), (0;0.2),(0.2;0.2) and (0.2,0) [press Apply (3) to validate each point]
3. Choose Sketch closure (4), Sketch validation (5)
4. Adjust zoom to view all objects (6)

Create Geometry: Points



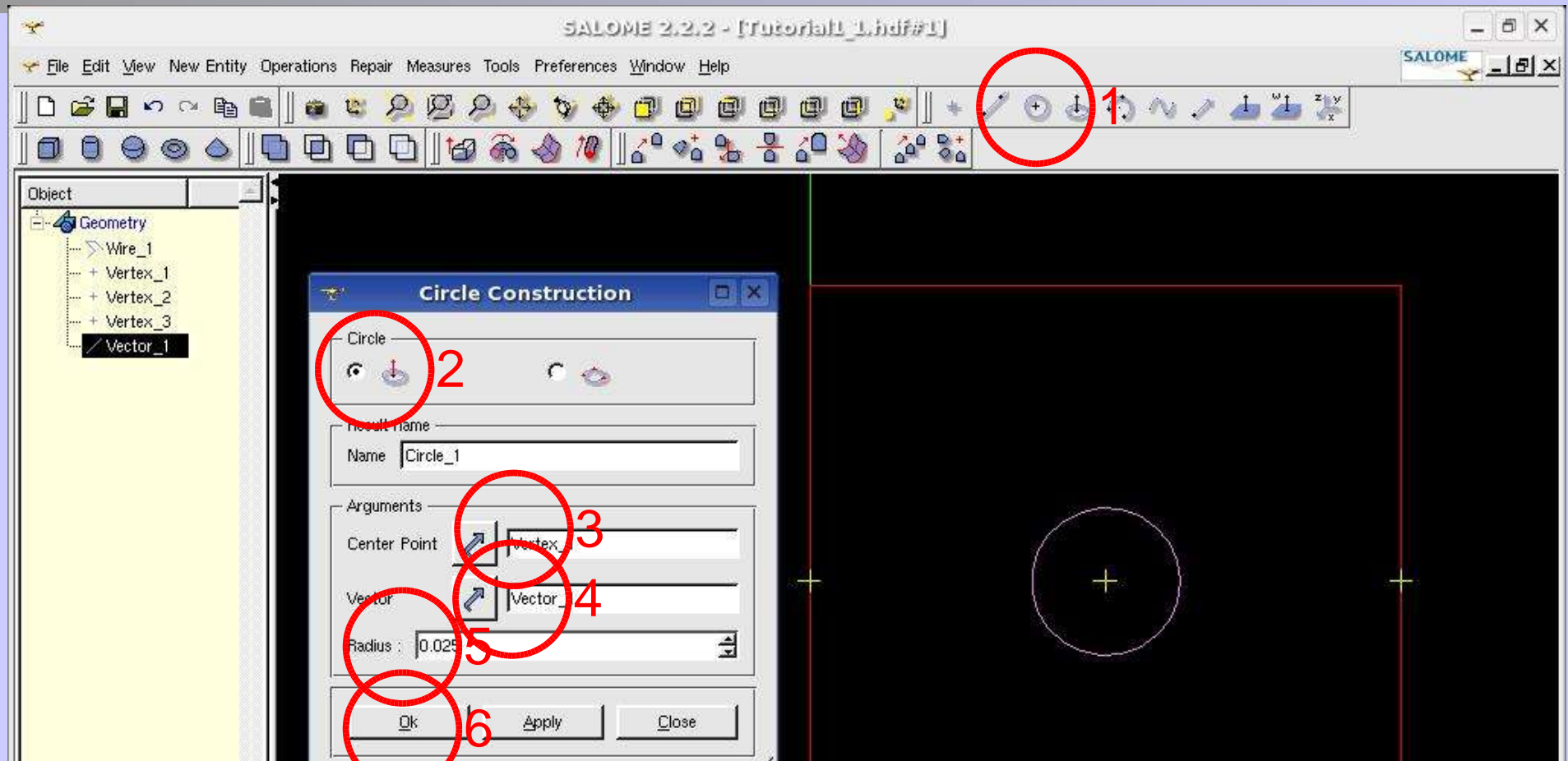
1. Choose Create Point
2. In Point Construction: choose Absolute Coordinates
3. Enter the following coordinates (0.1;0.1;0)
4. Press OK

Create Geometry: Vector



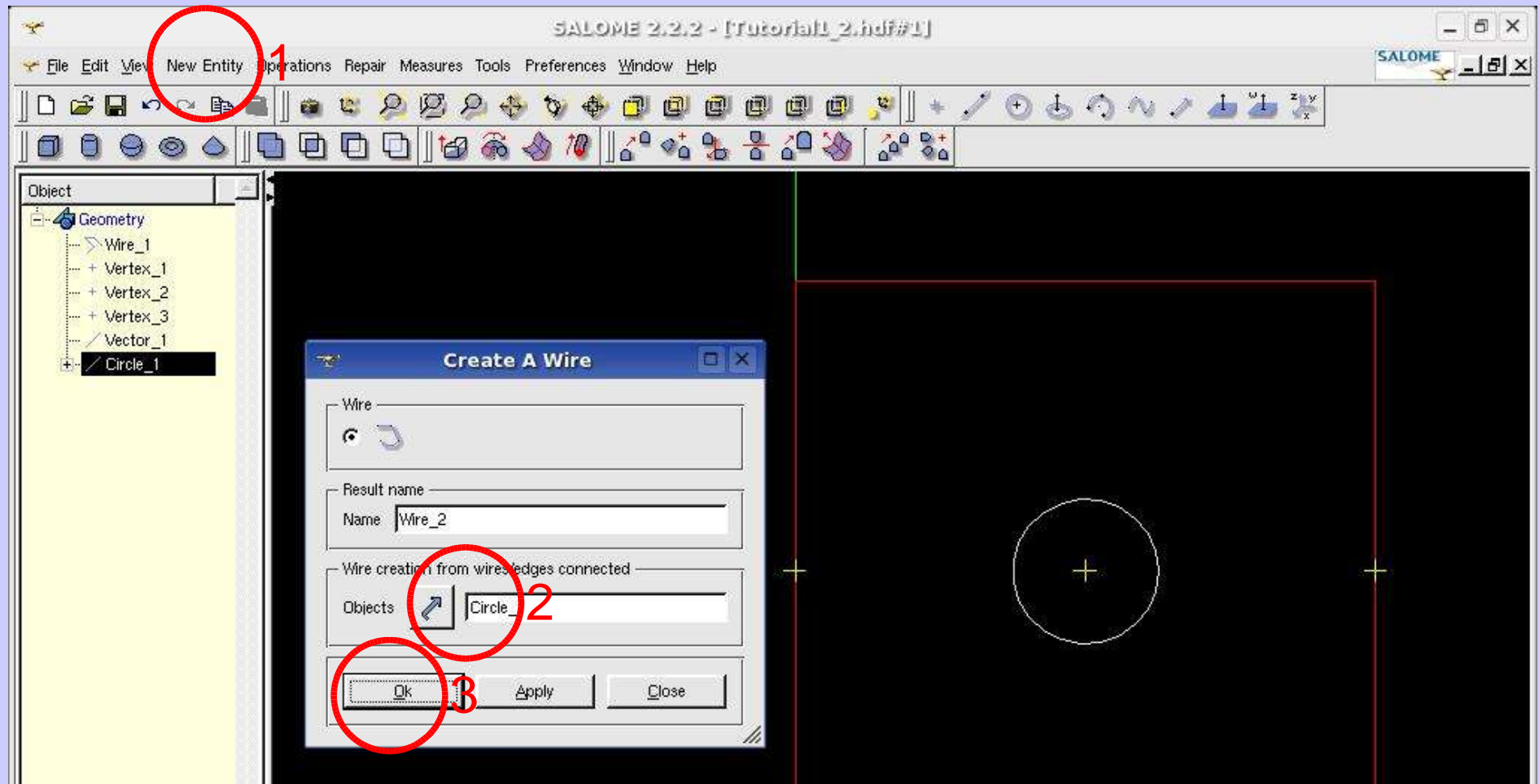
1. Choose Create Vector
2. In Vector Construction: choose option Components
3. Enter the following components (0;0;1)
4. Press OK

Create Geometry: Circle



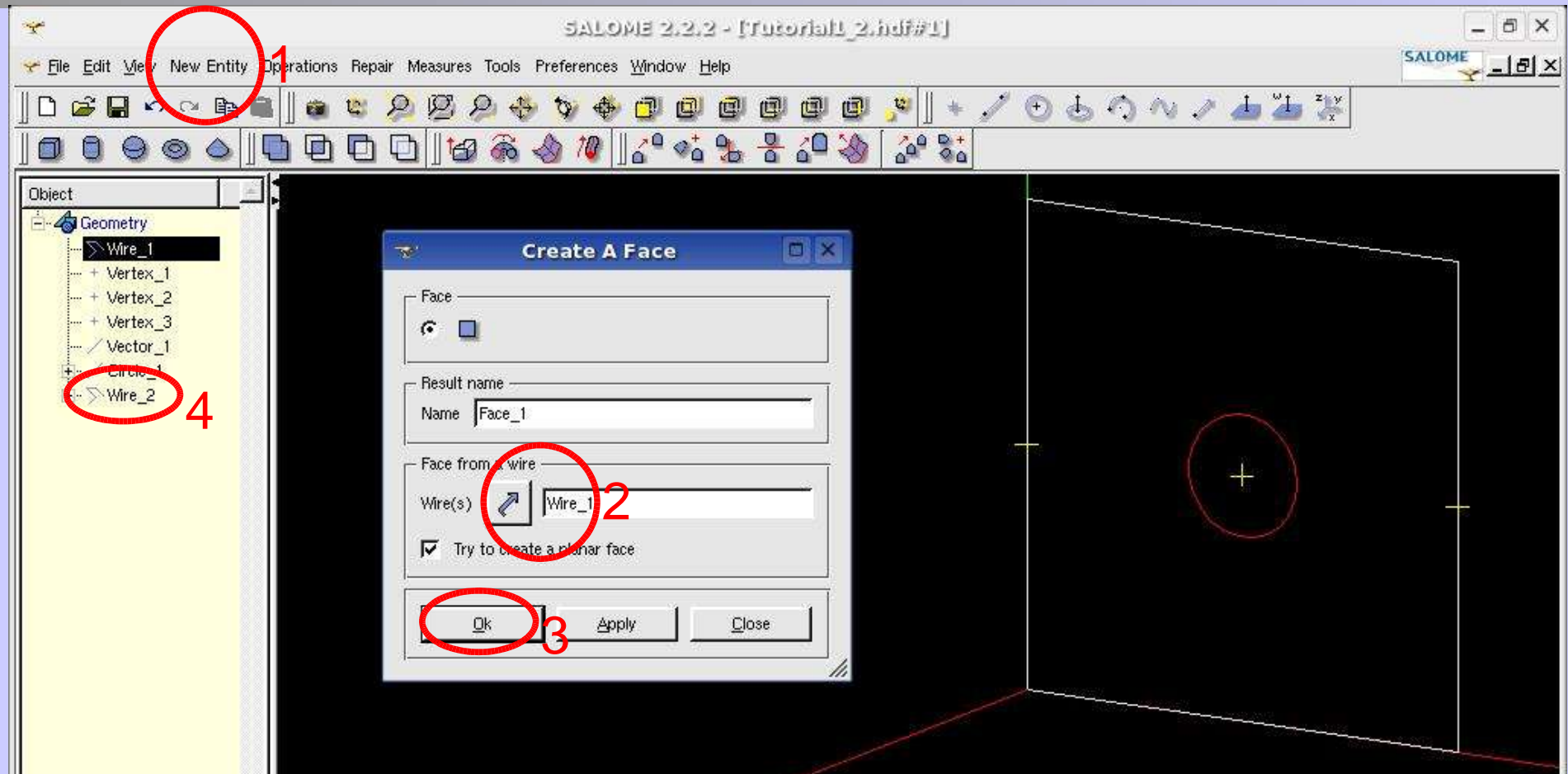
1. Choose Create Circle (1) with By "Point&Vector" option (2)
2. Select Center Point:(3): first click the Arrow button, then select Vertex_1 in either the 3D view or the tree view.
3. Select Vector (4): click Arrow and select Vector_1 in tree view.
4. Enter Radius= 0.025 (5) and click Ok (6)

Create Geometry: Wire



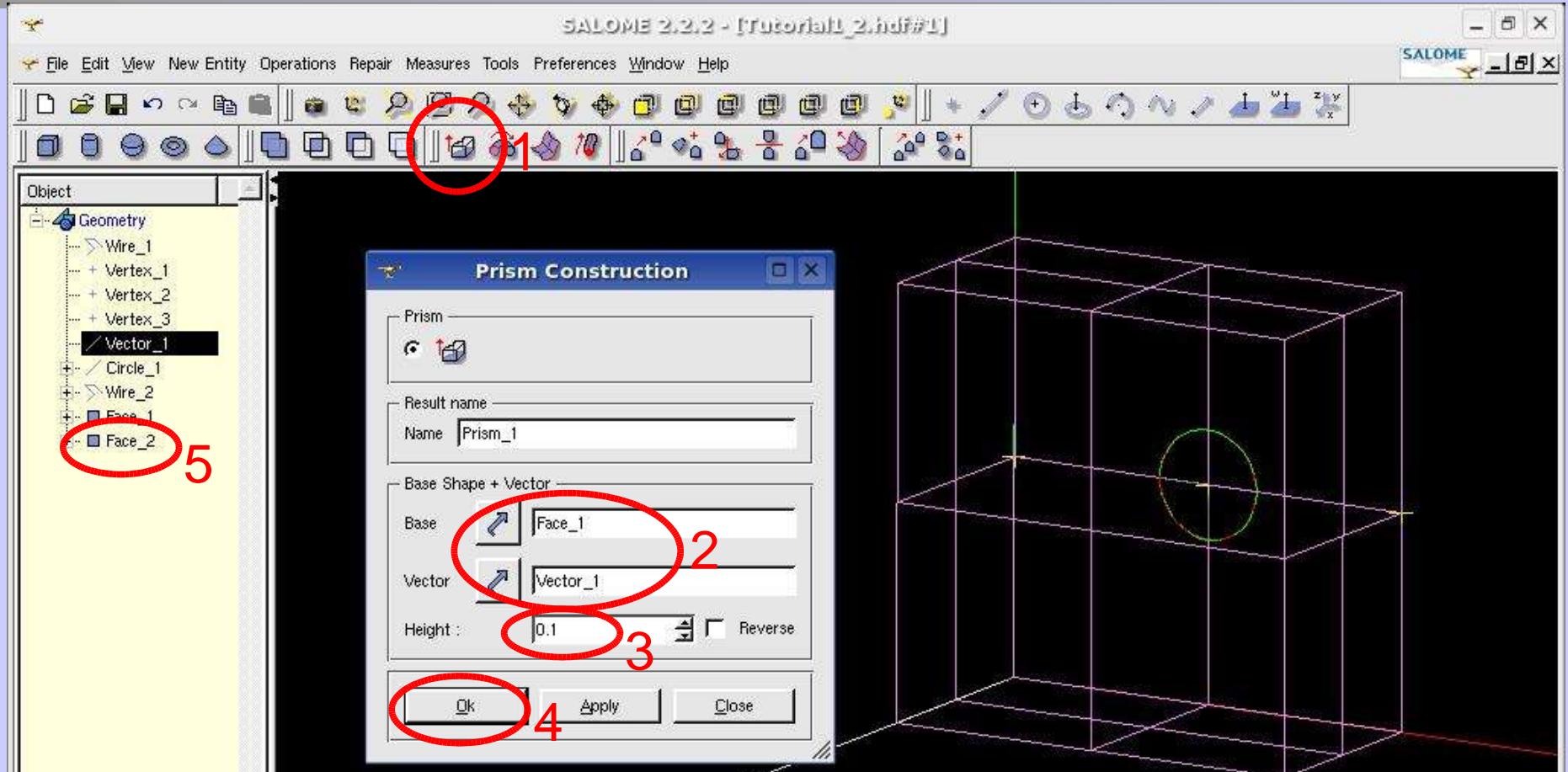
1. Choose New Entity-> Build-> Wire (1)
2. Select Objects (2): click "Arrow" and select Circle_1
[if needed select multiple objects by holding the shift key]
3. Click OK

Create Geometry: Faces



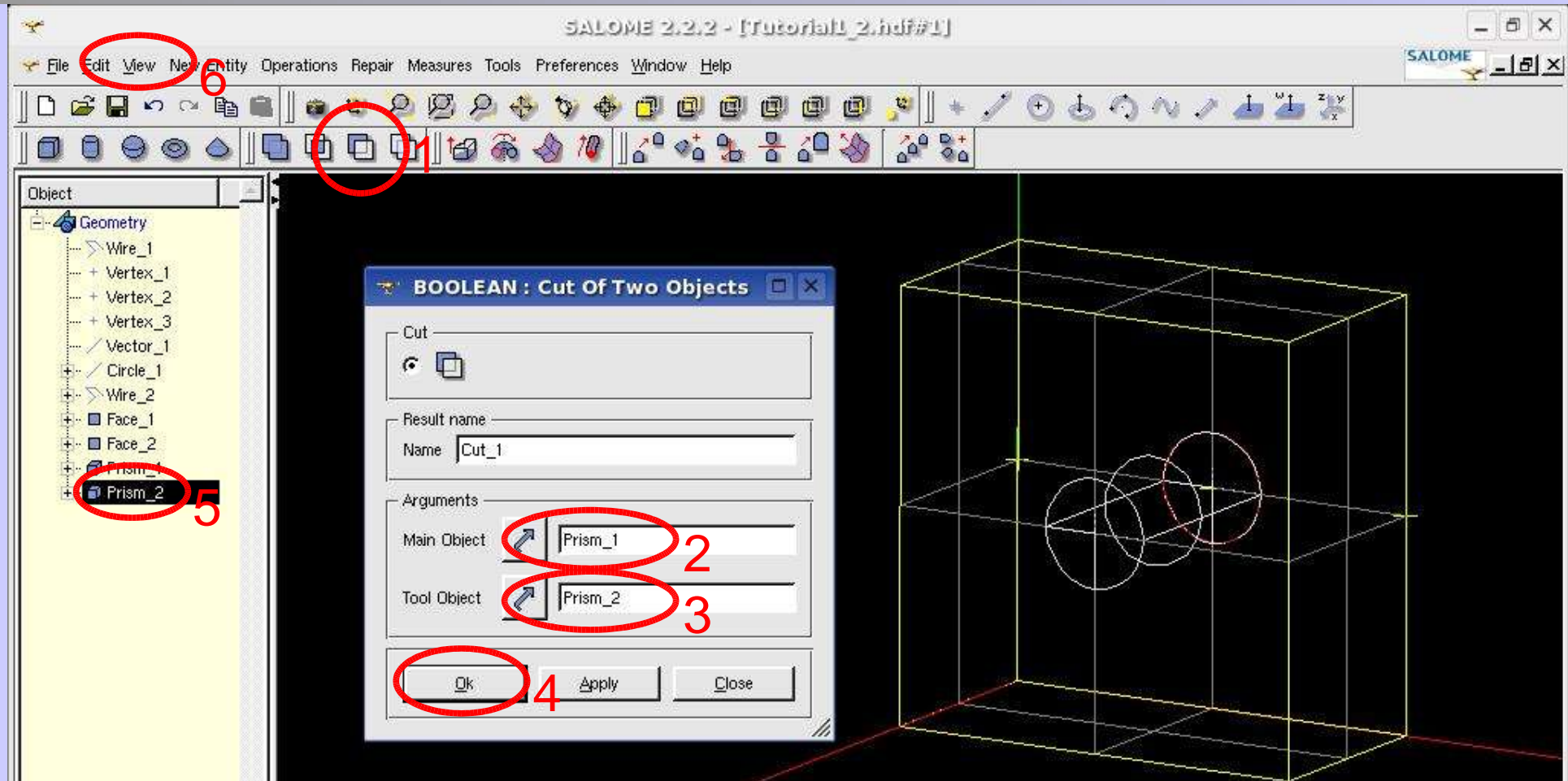
1. Choose New Entity-> Build-> Face (1)
2. Select wire (2): click "Arrow" and select Wire_1, click OK (3)
3. Repeat steps 1-2 for Wire_2 (4)
4. You can interactively zoom, pan & rotate by holding CTRL key with resp. Left, Middle & Right mouse button

Create Geometry: Extrusions



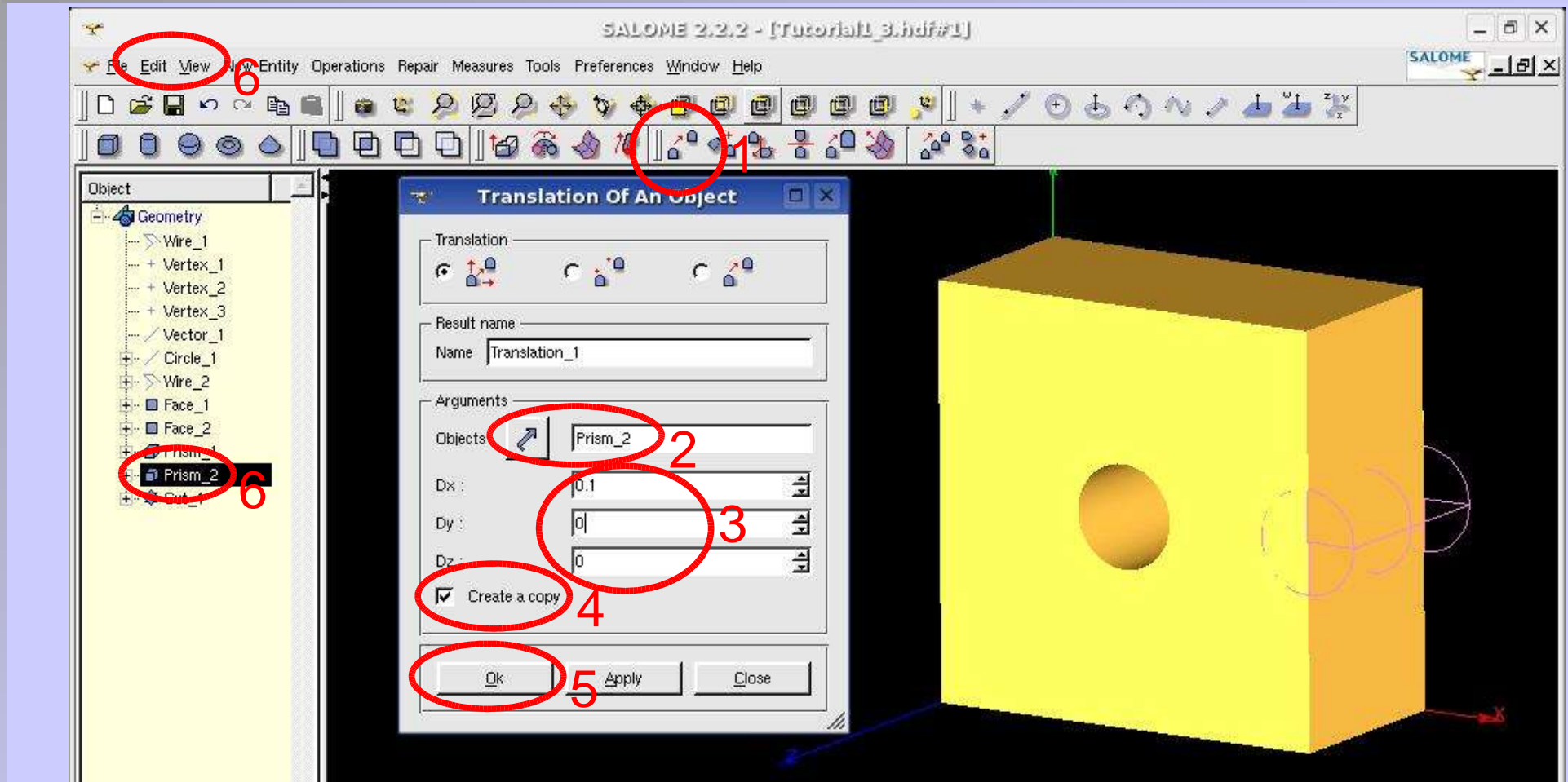
1. Choose Create Object by Extrusion (1)
- 2a. Select Base (2): click "Arrow" and select Face_1
- 2b. Select Vector (2): click "Arrow" and then select Vector_1 in tree view
3. Enter Height= 0.1 (3) and press OK (4)
4. Repeat steps 1-3 for Face_2 to create a cylinder (5)

Create Geometry: Boolean



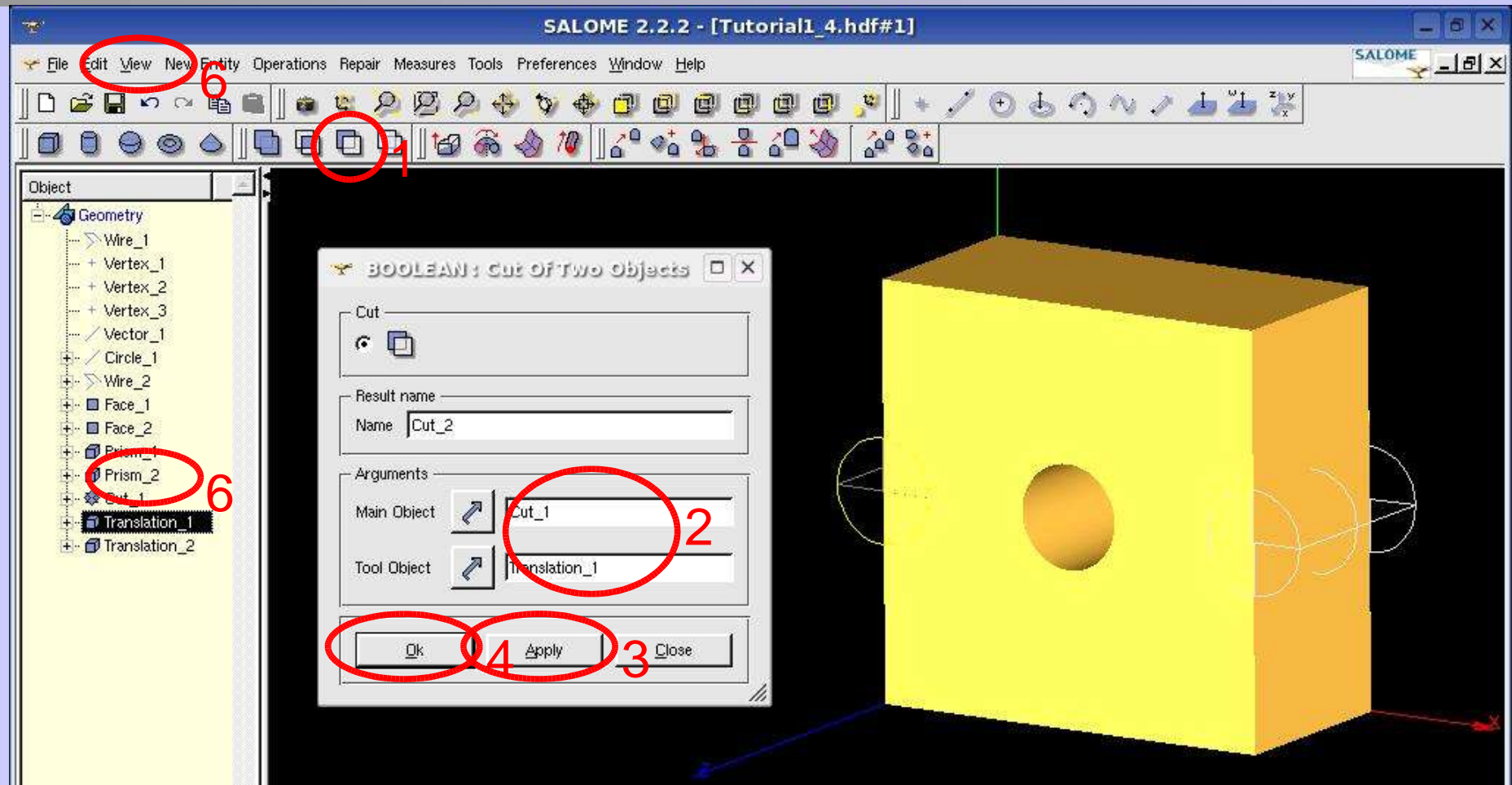
1. Choose Cut boolean operations (1)
2. Select Main object (2): select Prism_1
3. Select Tool object (3): select Prism_2 and click OK (4)
4. In tree view right click Cut_1 and choose Display Only (5)
5. Choose View-> Display mode-> Shading (6)

Create Geometry: Copy Features



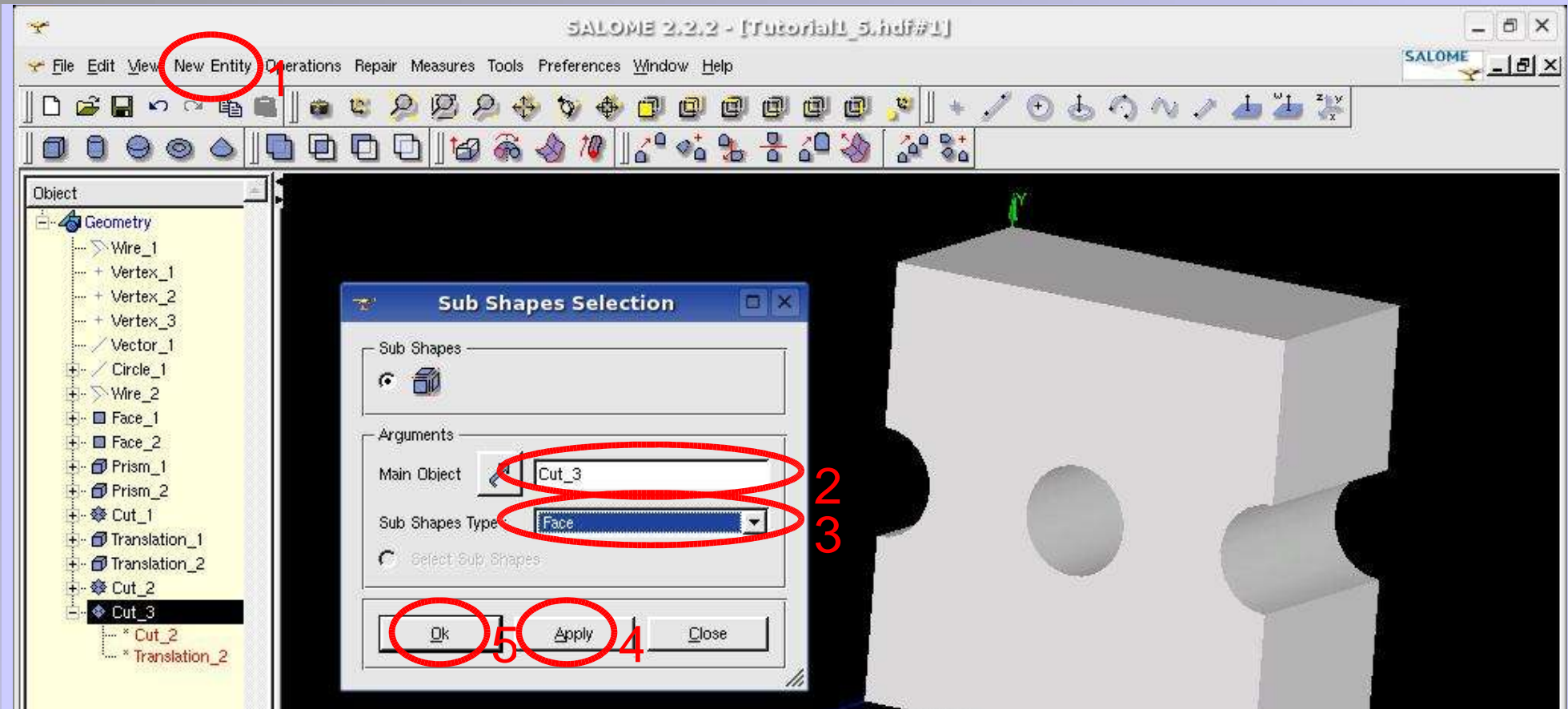
1. Choose Translate tool (1)
2. Select Object (2): select Prism_2
3. Translation vector (3): enter (0.1;0;0) and click OK (4)
4. Check Create Copy (4) and validate (5)
5. Repeat steps 1-4 with translation vector = $(-0.1;0;0)$

Create Geometry: new Cuts



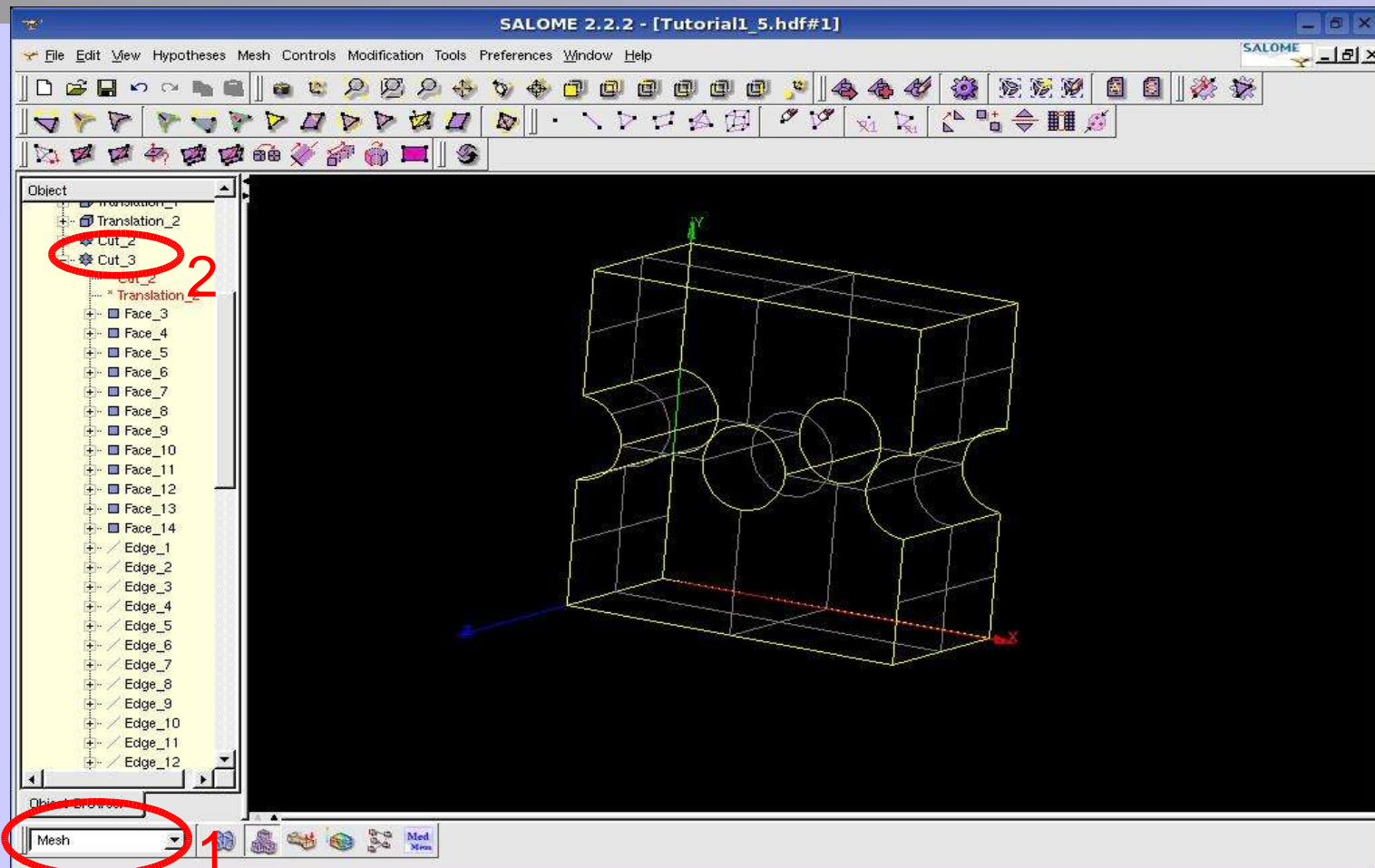
1. Choose Boolean Cut tool (1)
2. Set Main obj.= Cut_1 and Tool obj.= Translation_1 (2)
3. Click Apply (3)
4. Set Main obj.= Cut_2 and Tool obj.= Translation_2 (2)
5. Click OK (4)

Create Geometry: Explode



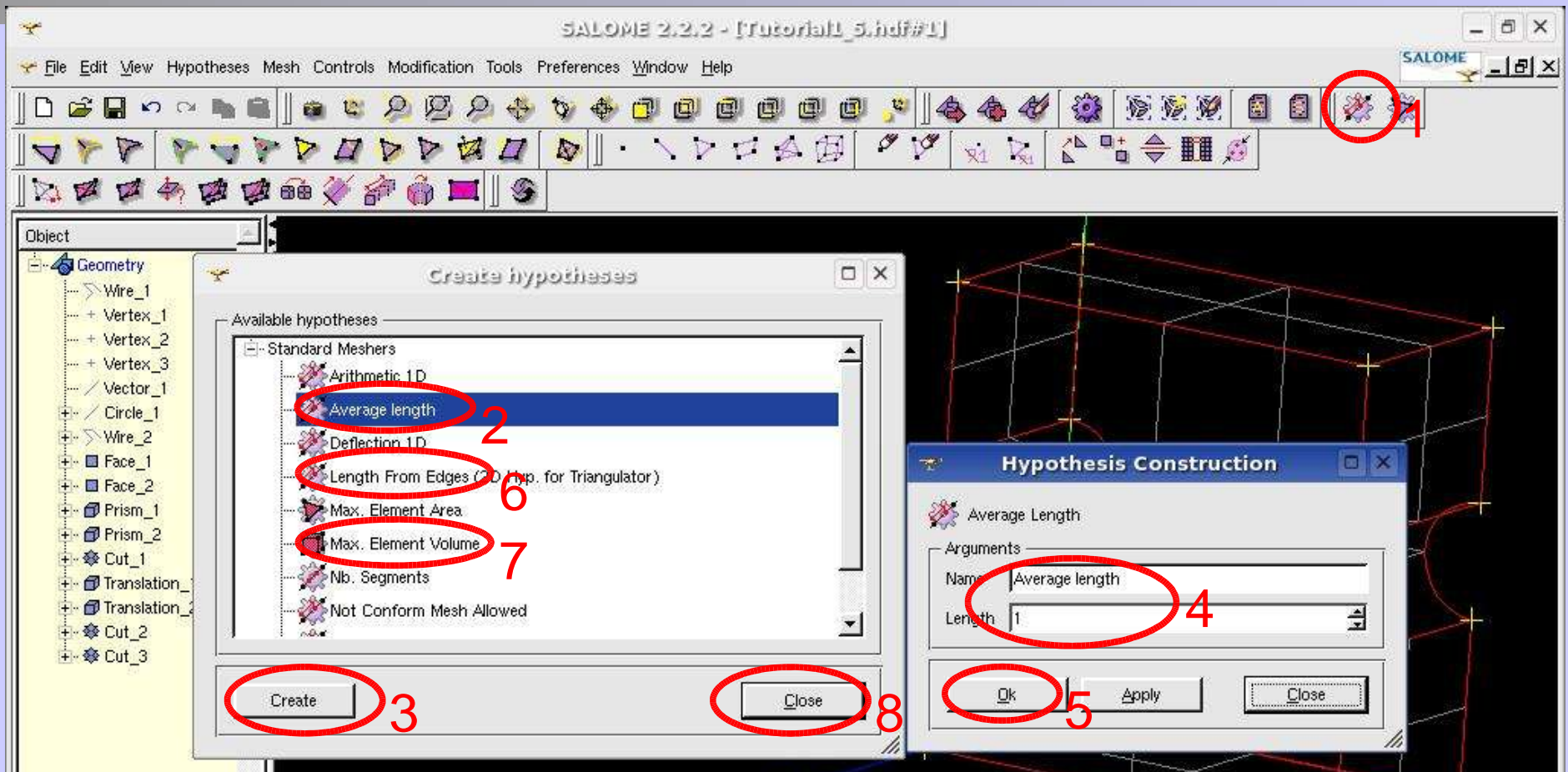
1. Choose New Entity-> Explode (1)
2. Select Cut_3 (2), choose Shape type= Face(3), click Apply(4)
3. choose Shape type= Edge (3) ,click Apply (4)
4. finally, choose Shape type= Vertex (3) and click OK (5)
5. Note that now you can see, select & use the extracted objects by selecting them in the tree view or the 3D view.

Mesh: Display geometry



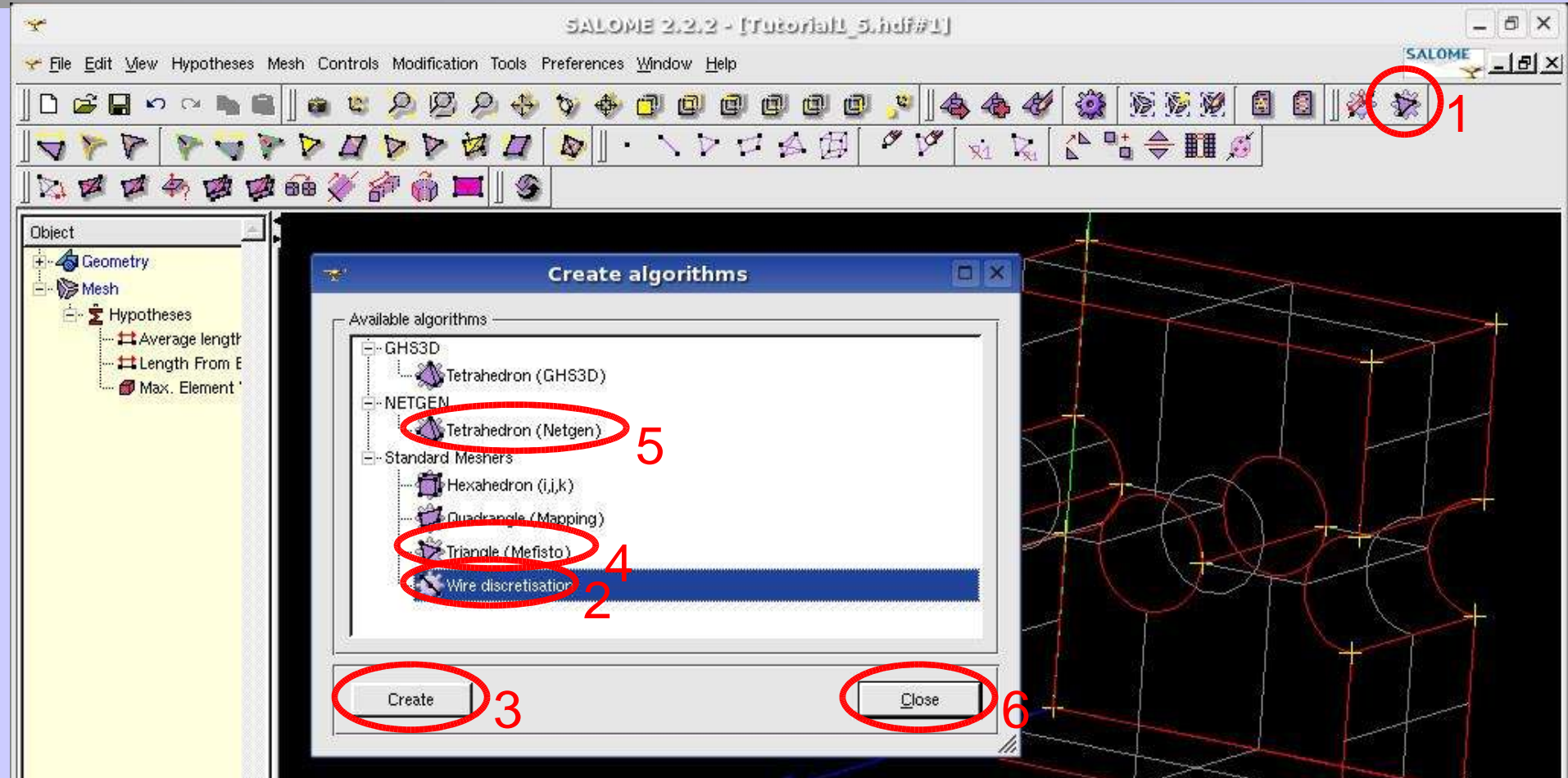
1. Choose Mesh module (1), expand the tree of Cut_3 (2)
2. In tree view: select first Face item, hold the shift key and then click on the last Edge item to select all the Face/Edge/Vertex items of Cut_3
3. Right click on selection in tree view, choose Display Only

Mesh: hypotheses



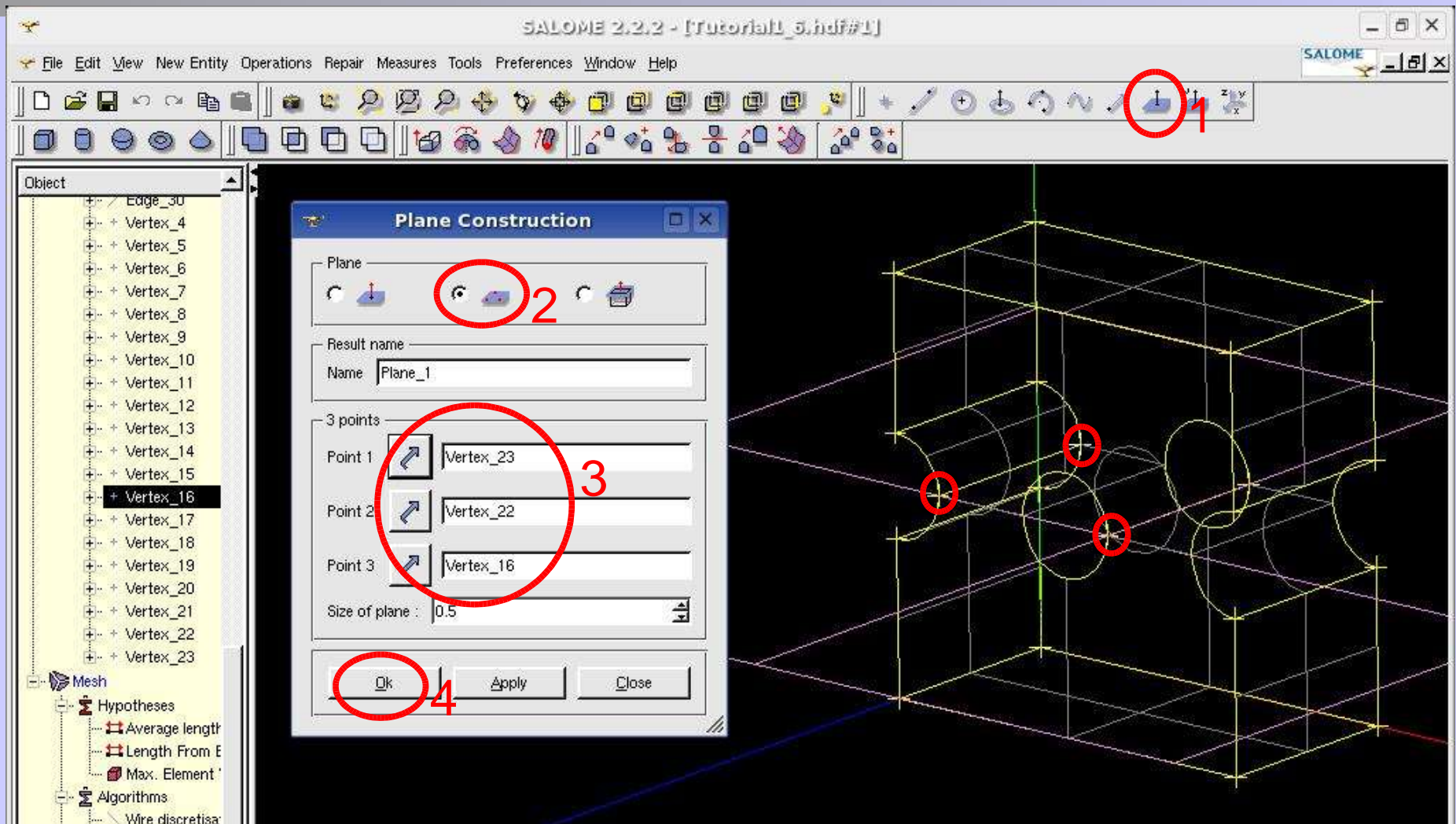
1. Click Create Hypotheses (1), choose "Average Length" (2), click Create (3), then set Length= 0.02 (4) and click Ok (5)
3. Select "Length from Edges" (6) and click Create (3)
4. Select "Max. Element Volume" (7), click Create (3), set Max Volume= 0.01 (4) and click Ok (5).
5. Finally close hypotheses dialog (8)

Mesh: algorithms



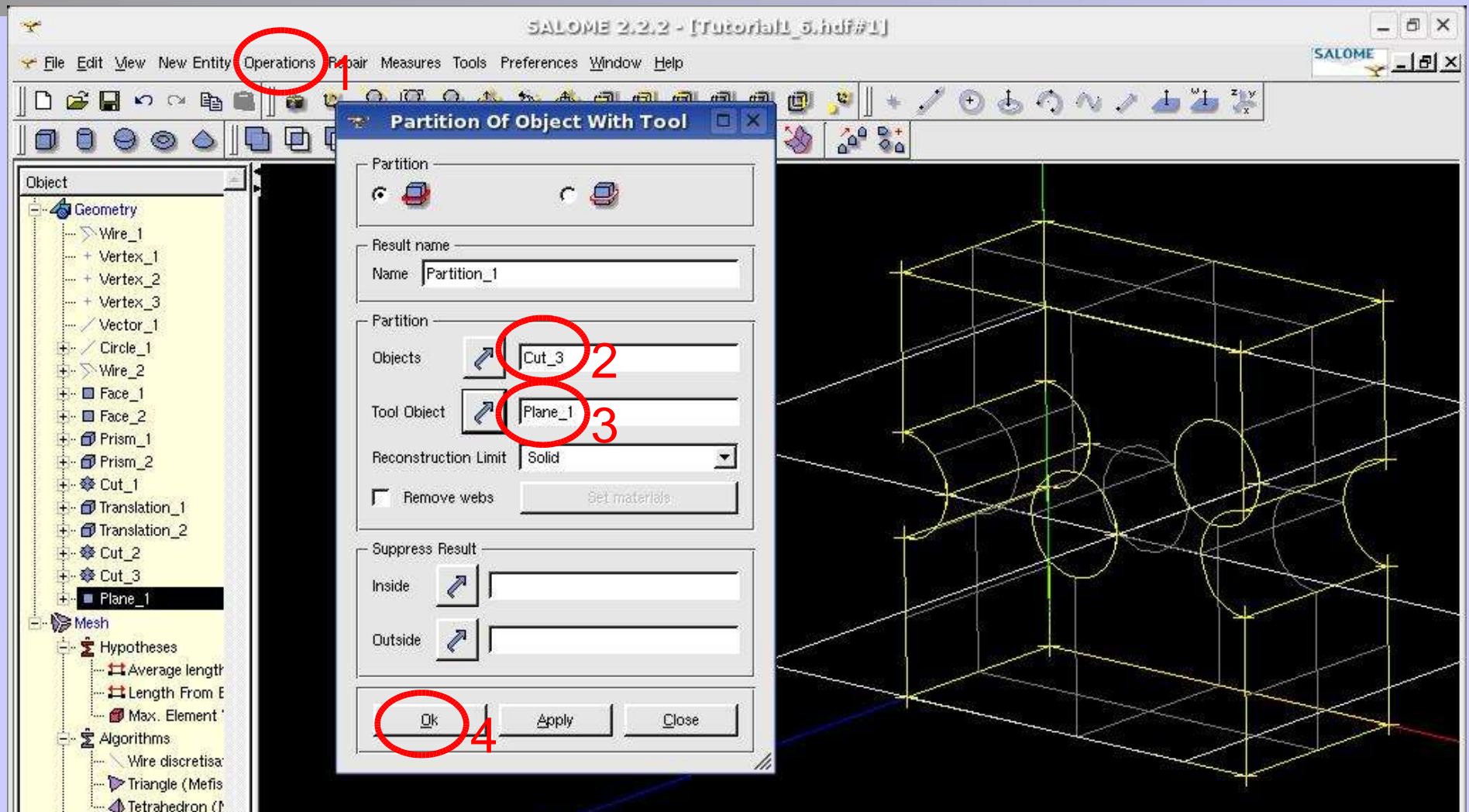
1. Click Create Algorithm (1), choose "Standard / Wire discretisation" (2) and click Create (3)
3. Select "Standard / Triangle" (4) and click Create (3)
4. Select "Netgen / Tetrahedron" (5), click Create (3),
5. Finally close Algorithm dialog (6)

Mesh: Partitioning plane



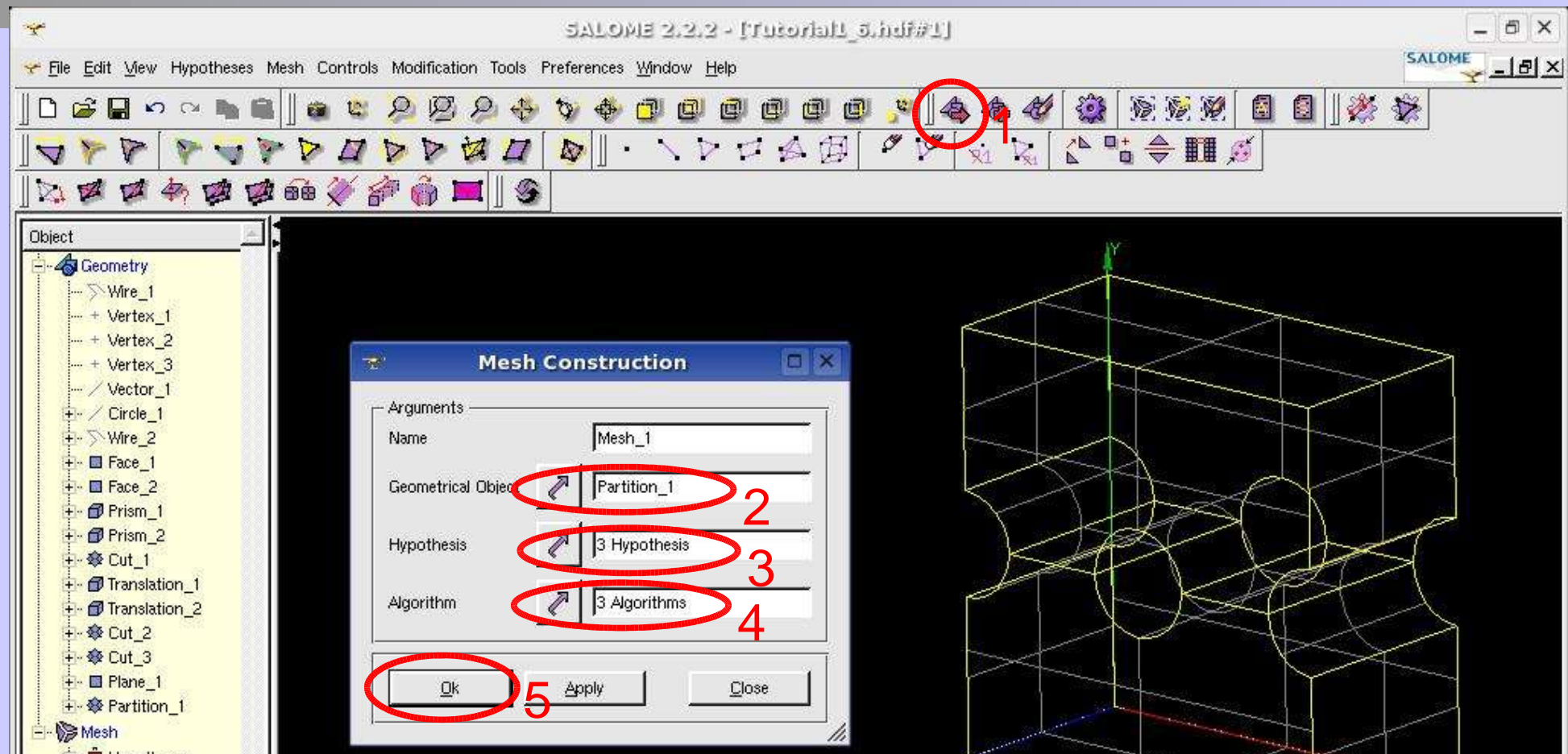
1. Go back to Geom module and click Create Plane (1)
2. Select Plane by 3 points (2), select the highlighted vertices (3) and press OK (4)

Mesh: Partition



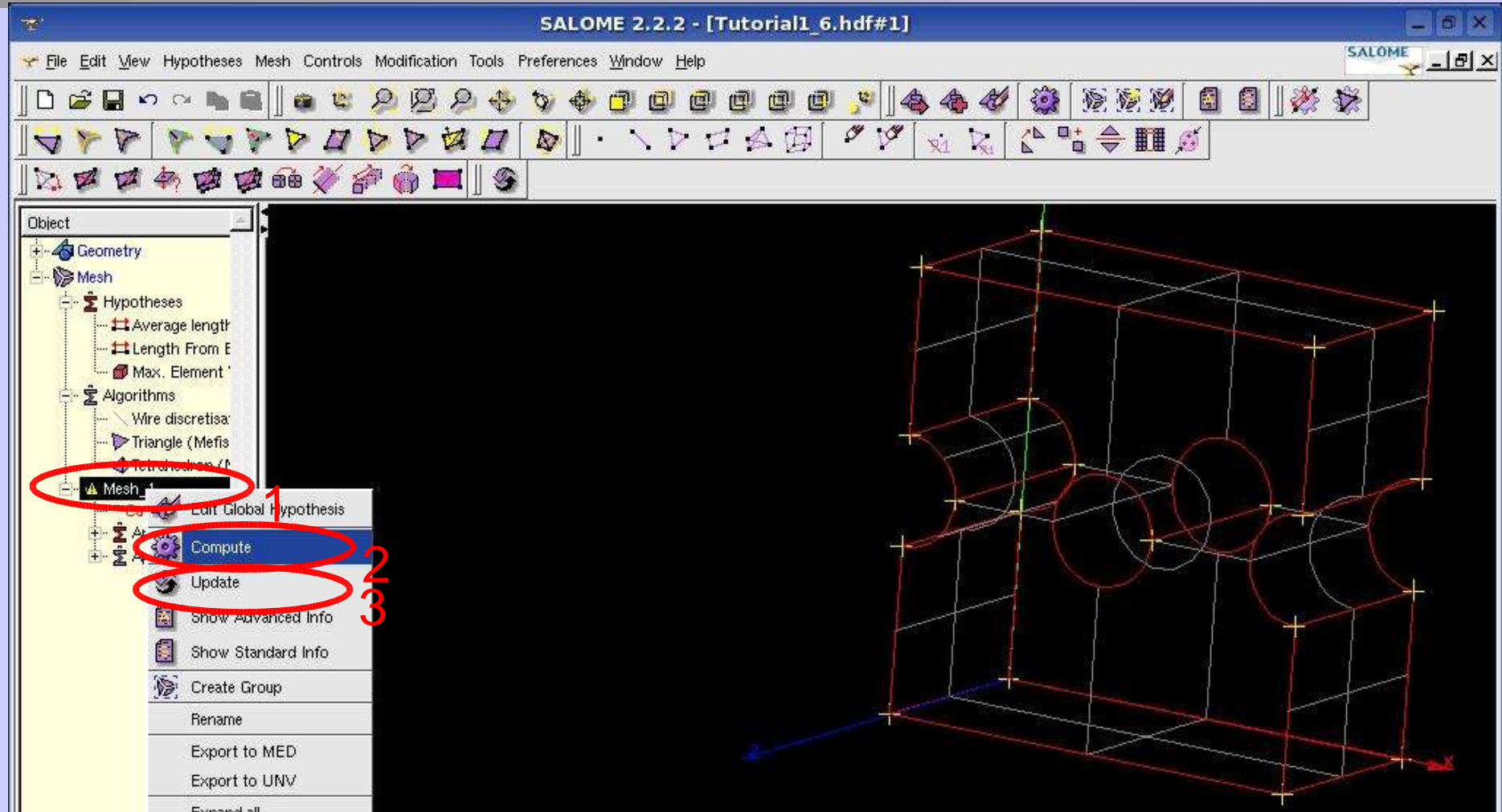
1. Choose Operations-> Partition
2. Select Object Cut_3 (2), select Tool= Plane_1 (3) and press OK (4)
3. Return to Mesh Module

Mesh: Mesh Construction



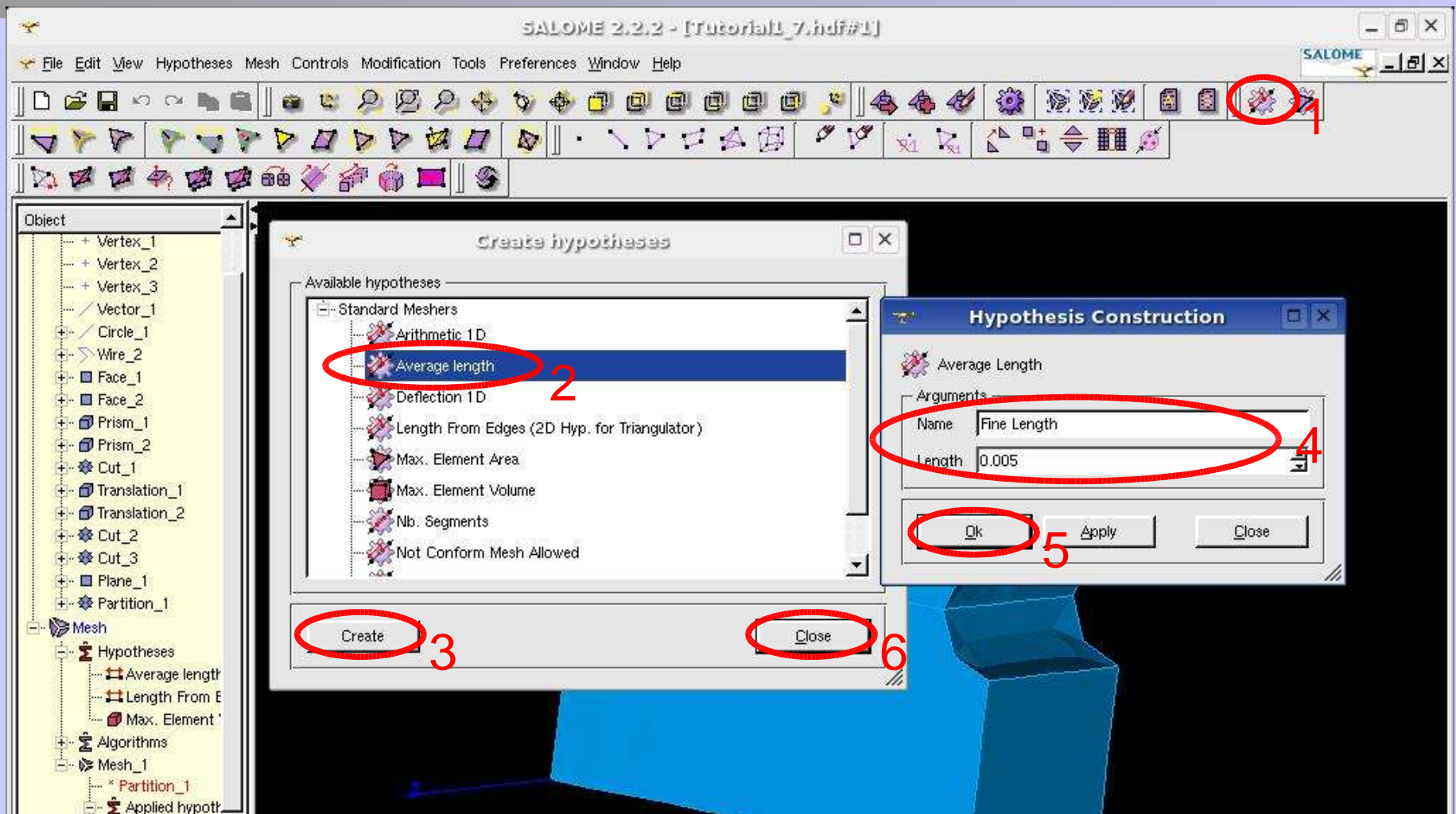
1. Click Global Hypotheses(1), select Geometry= Partition_1(2)
2. Select hypotheses (3): click 'arrow', and with shift key select all three hypotheses from tree view
3. Select algo. (4): click 'arrow', and with shift key select all three algorithms from tree view. Finally click OK (5)

Mesh: Mesh Construction



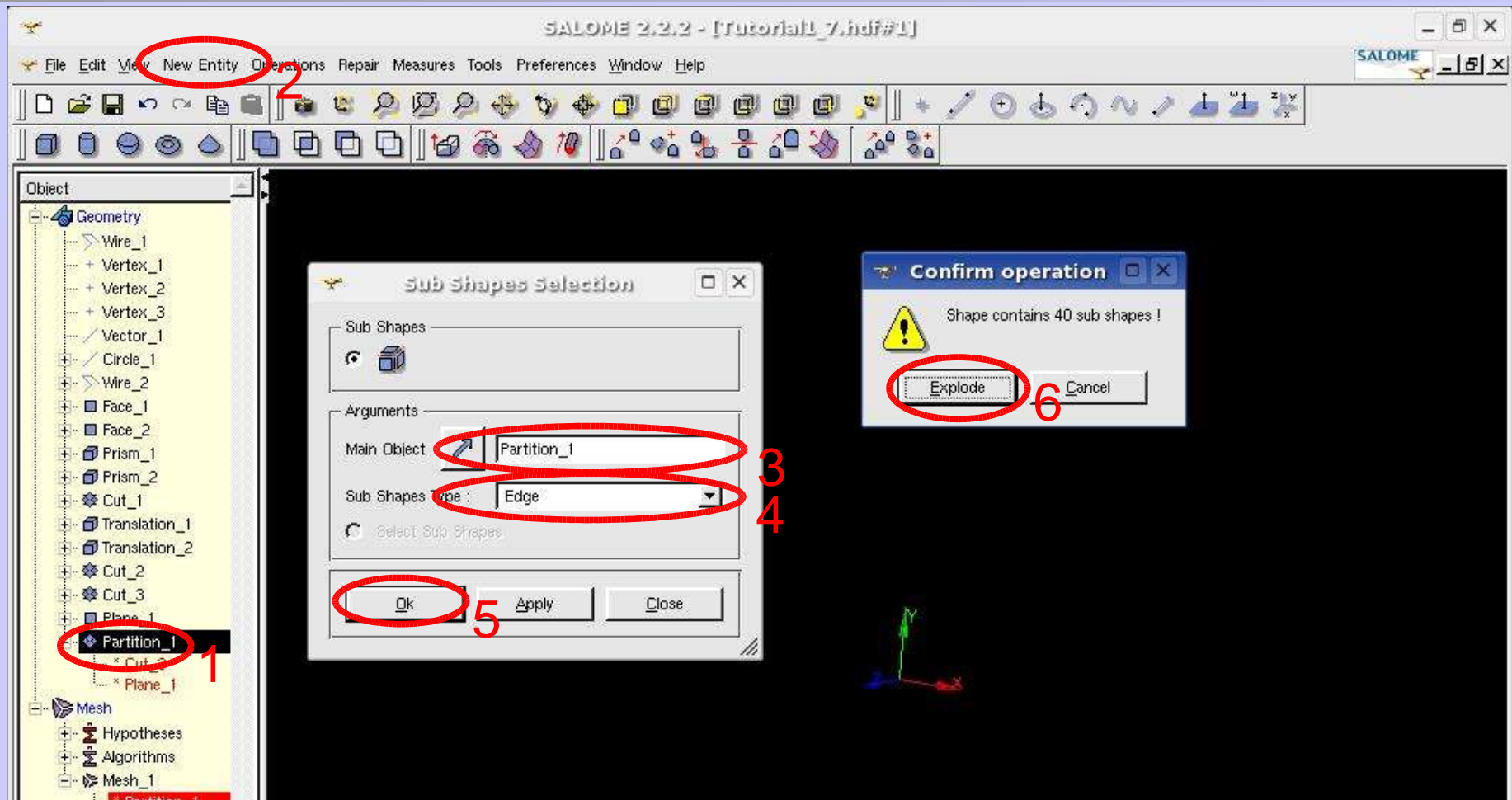
1. Right-click "Mesh_1" (1), select Compute to generate the mesh (2). Be patient, this can take a while...
2. When finished, right-click "Mesh_1" (1), select Update to display the new mesh (3)

Mesh: Mesh Refinement



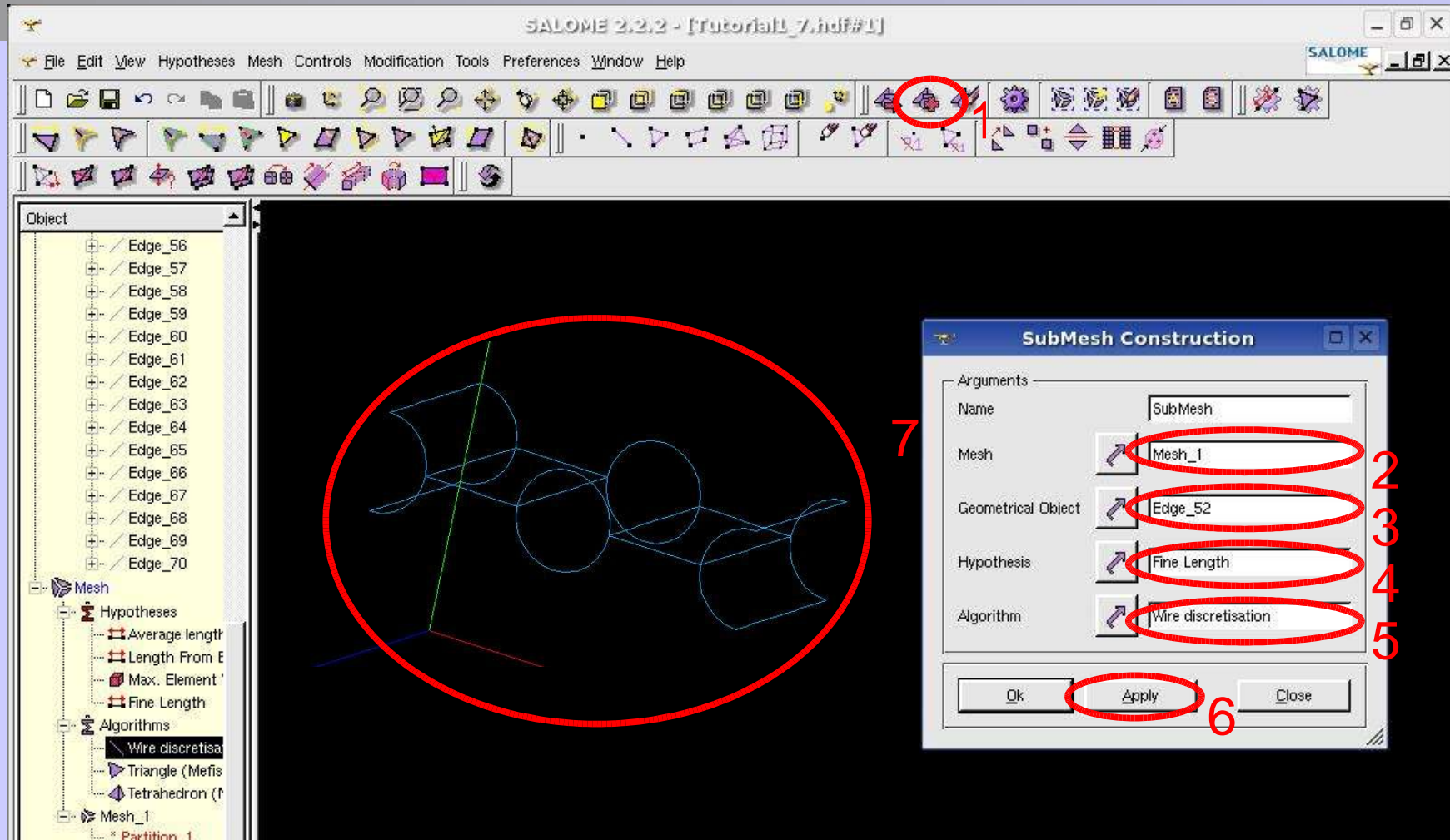
1. Click "Create Hypothesis" (1), select "Average Length" (2) & click "Create" (3)
2. Enter Name= "Fine Length" & Length= 0.007 (4), click Ok(5)
3. Click Close (6)

Mesh: Mesh Refinement



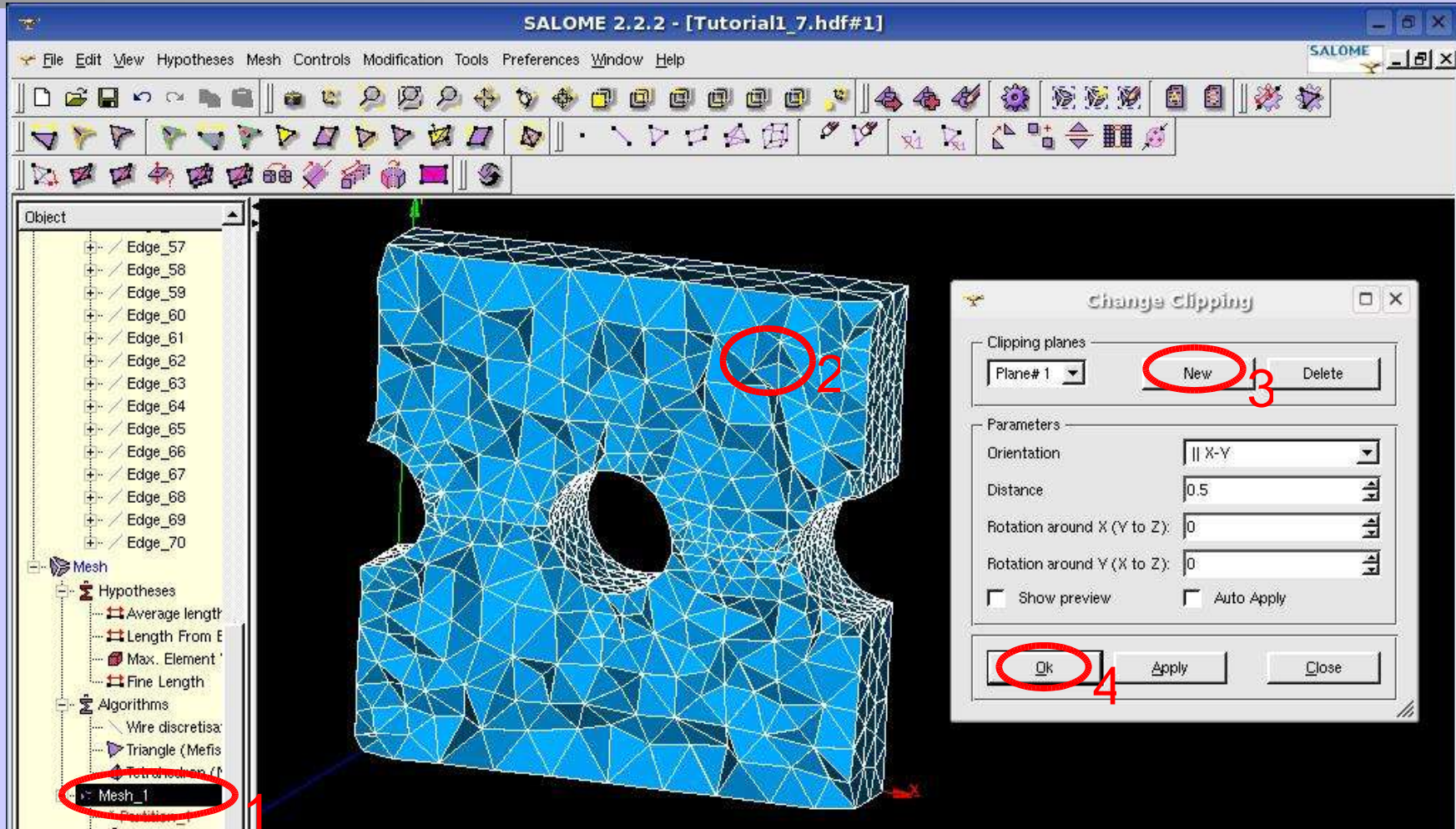
1. Switch to GEOM module. Right click Partition_1-> Erase (1)
2. Choose menu New Entity-> Explode.
3. Select Object= Partition_1 (3) & Shape type= Edge (4)
4. Click Ok (5) & Validate by clicking Explode (6).
5. Switch to MESH module.

Mesh: Mesh Refinement



1. Click "Create Local Hypothesis" (1)
2. Select Mesh= Mesh_1(2), Hypothesis= Fine Length(4), Algorithm= Wire distribution(5).
3. Click Arrow (3), for each entity of figure: select the object in 3D view (7) & validate by Apply (3). Close the dialog.

Mesh: Checking Mesh



1. Right click "Mesh_1" -> Compute (1).
2. When finished, right click "Mesh_1" -> Update (1)
3. Right click the mesh in 3D view (2), choose "Clipping"
4. Click "New" (3) & OK (4).
5. Right click the object (2) -> Control -> Aspect Ratio 3D