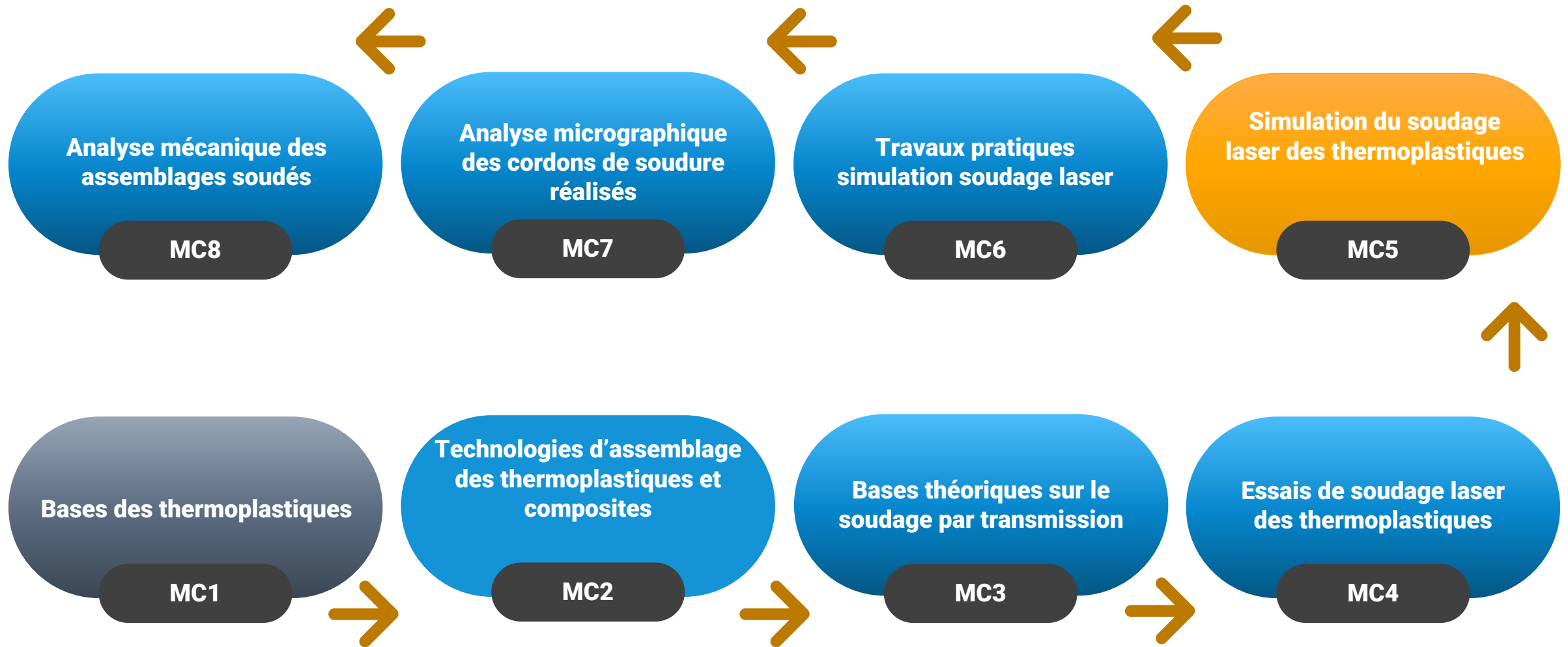


Technologie d'AssemBLage dEs ThermoplasTiques par LasEr

Micro-Contenus (MC)

Technologie d'Assemblage des Thermoplastiques par Laser



Objectif pédagogique final

MC5

**Acquérir les notions de base de la simulation
numérique par éléments finis
du soudage laser des thermoplastiques**

MC5 : Simulation du soudage laser des thermoplastiques

Module 7 **Evaluation**

Module 6 **Conclusion**

Module 5 **Analyse des résultats de simulation**

Module 4 **Simulation numérique du champ de température pendant le soudage laser (Comsol Multiphysics)**

Module 3 **Généralités sur les outils de modélisation thermique du soudage laser**

Module 2 **C'est quoi une simulation numérique par éléments finis?**

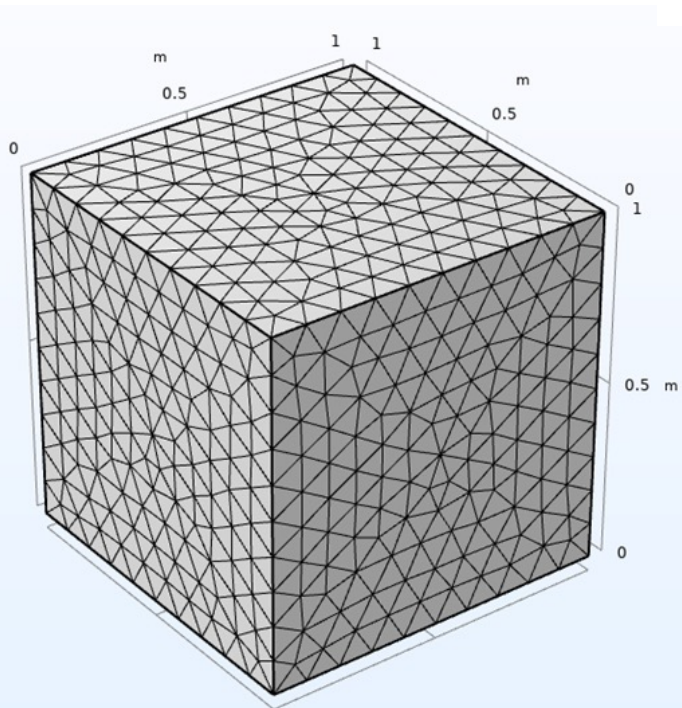
Module 1 **Introduction générale**

A la fin de ce micro-contenu, vous **serez capable** de :

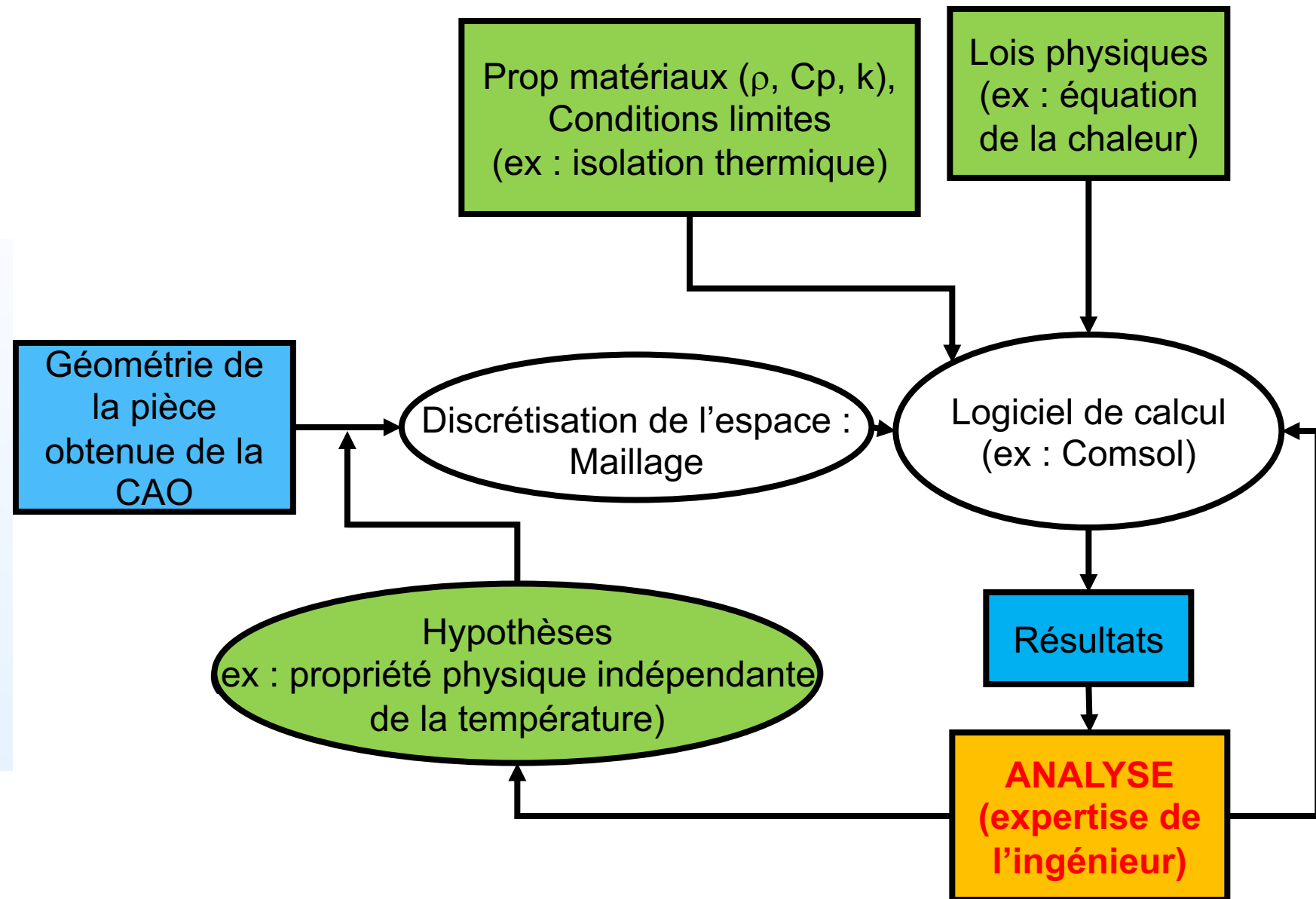
- Expliquer** les différentes conditions aux limites pour la simulation numérique du soudage laser.
- Exploiter** les résultats issus d'un calcul éléments finis.
- Expliquer** les différentes étapes de base d'un calcul par éléments finis.
- Construire** un modèle simple de calcul thermique.

**C'est quoi une simulation numérique
par éléments finis?**

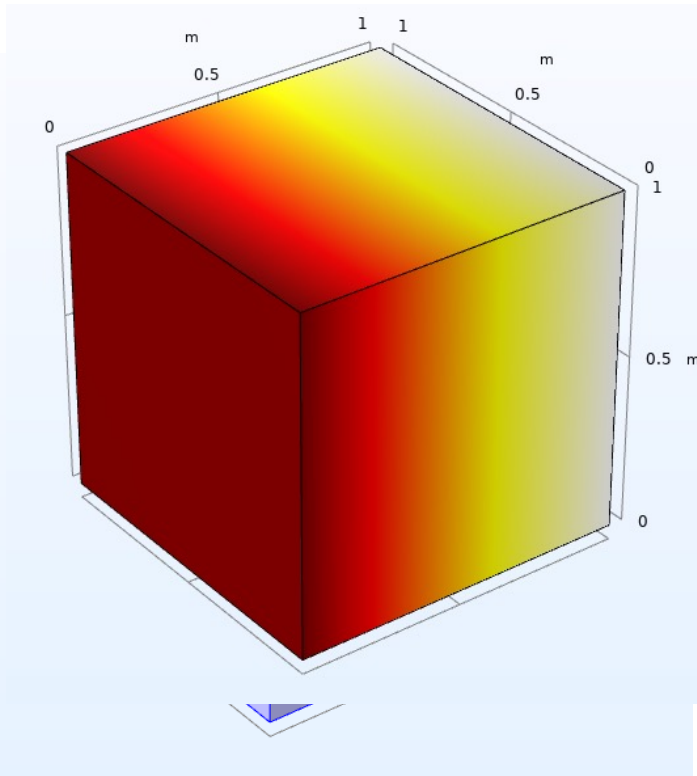
M2 – C'est quoi une simulation numérique par éléments finis?



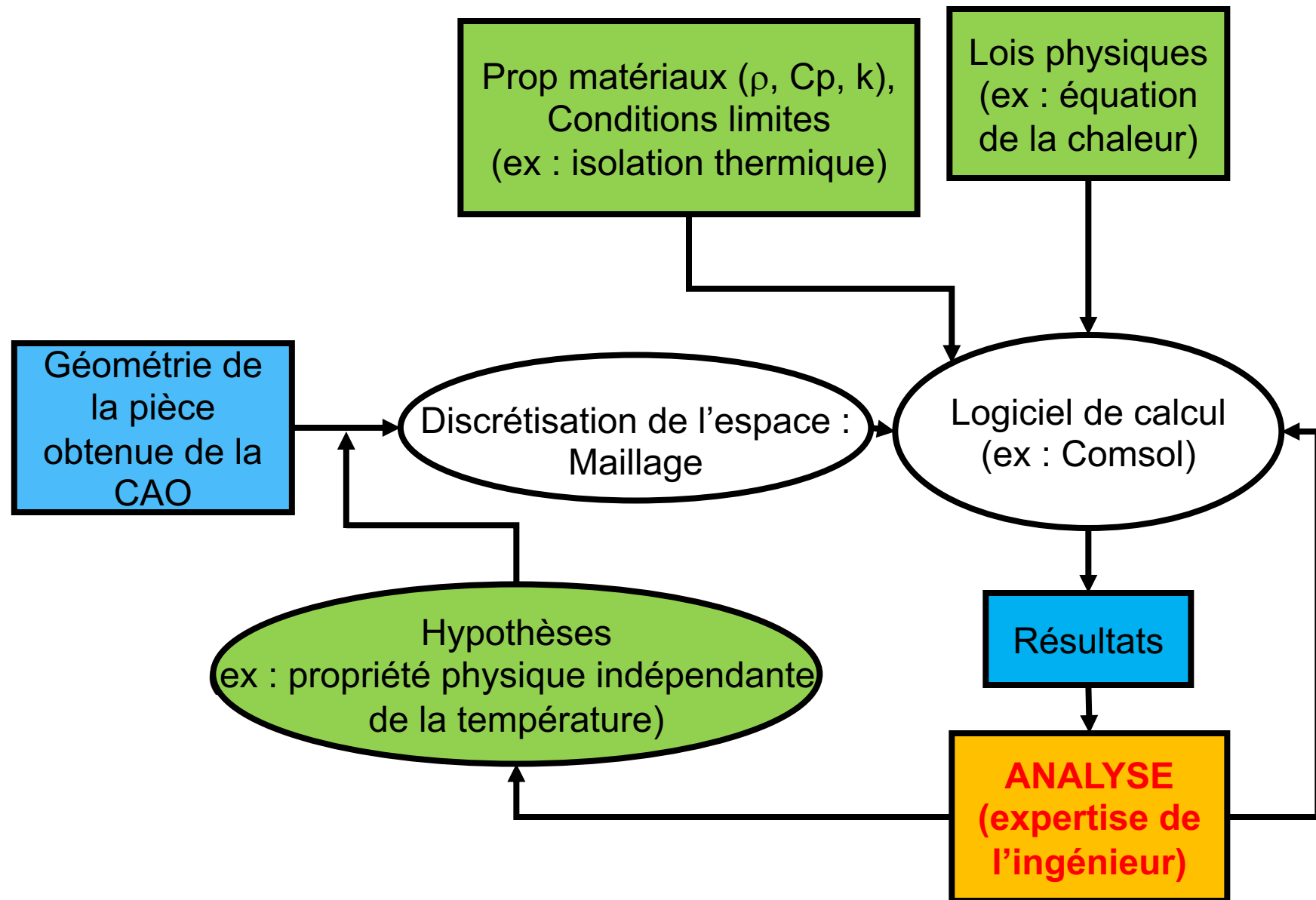
Maillage du domaine



M2 – C'est quoi une simulation numérique par éléments finis?



Résultats de calcul



**Généralités sur les outils de
modélisation thermique du soudage
laser des composites**

M3 – Généralités sur les outils de modélisation thermique du soudage laser

The screenshot displays the COMSOL Multiphysics software interface. The top ribbon includes tabs for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. The main workspace is divided into three panels: Model Builder, Settings, and Graphics.

Model Builder: Shows a tree view of the model structure. The selected study is "Study 1" with a time-dependent step. The physics interface is "Heat Transfer in Solids (ht)".

Settings: The "Heat Transfer in Solids" settings are shown. The domain selection is set to "All domains". The equation form is "Study controlled" and the equation is assumed to be "Study 1, Time Dependent". The governing equation is displayed as:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{red}}$$
$$\mathbf{q} = -k \nabla T$$

The reference temperature is set to "User defined" with a value of 293.15 [K].

Graphics: A 3D model of a cube is shown in the graphics window. The cube is colored in a gradient from light blue to dark blue. The dimensions are indicated as 1 m for the length and width, and 0.5 m for the height. A coordinate system (x, y, z) is visible at the bottom left of the graphics window.

Messages: The bottom panel shows the Messages window with the following text:

```
COMSOL Multiphysics 5.6.0.401
[Jun 20, 2022 3:52 PM] Finalized geometry has 1 domain, 6 boundaries, 12 edges, and 8 vertices.
```

M3 – Généralités sur les outils de modélisation thermique du soudage laser

The screenshot displays the COMSOL Multiphysics software interface. The top menu bar includes File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. The ribbon contains various toolbars for Build, Import/Export, Cleanup, Primitives, Work Plane, Operations, and Other.

The **Model Builder** pane on the left shows a tree view of the model structure:

- Untitled.mph (root)
 - Global Definitions
 - Parameters 1
 - Materials
 - Component 1 (comp 1)
 - Definitions
 - Geometry 1
 - Block 1 (blk 1)
 - Form Union (fin)
 - Materials
 - Heat Transfer in Solids (ht)
 - Solid 1
 - Initial Values 1
 - Thermal Insulation 1
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Results
 - Datasets
 - Derived Values
 - Tables
 - Export
 - Reports

Settings

Mesh

Build All

Label: Mesh 1

Mesh Settings

Sequence type: Physics-controlled mesh

Physics-Controlled Mesh

Element size: Normal

Contributor: Heat Transfer in Solids (ht) Use

Graphics

The graphics window shows a 3D view of a cube mesh. The dimensions are indicated as 0.5 m and 1 m. A coordinate system with x, y, and z axes is visible at the bottom left.

Messages | Progress | Log | Table

COMSOL Multiphysics 5.6.0.401
[Jun 20, 2022 3:52 PM] Finalized geometry has 1 domain, 6 boundaries, 12 edges, and 8 vertices.
[Jun 20, 2022 3:52 PM] Complete mesh consists of 16547 domain elements, 1536 boundary elements, and 120 edge elements.

M3 – Généralités sur les outils de modélisation thermique du soudage laser

The screenshot displays the COMSOL Multiphysics interface for a thermal simulation. The main window shows a 3D plot of a cube with a temperature distribution. The plot is titled "Surface: Temperature (K)" and features a color scale on the right ranging from 294 K (dark red) to 312 K (yellow). The cube's dimensions are indicated as 1 m on the top edges and 0.5 m on the side edges. The plot is viewed from an isometric perspective.

The interface includes several panels:

- Model Builder:** Shows the hierarchy of the model, including "Study 1" and "Solution 1 (sol1)".
- Settings:** Configures the 3D plot, including the label "Temperature (ht)", dataset "Study 1/Solution 1 (sol1)", and plot settings like "View: Automatic" and "Color: From theme".
- Graphics:** Contains the 3D plot and a toolbar for navigation and visualization.
- Messages:** Displays simulation logs, including the following text:
COMSOL Multiphysics 5.6.0.401
[Jun 20, 2022 3:52 PM] Finalized geometry has 1 domain, 6 boundaries, 12 edges, and 8 vertices.
[Jun 20, 2022 3:52 PM] Complete mesh consists of 16547 domain elements, 1536 boundary elements, and 120 edge elements.
[Jun 20, 2022 3:53 PM] Number of degrees of freedom solved for: 23646 (plus 19865 internal DOFs).
[Jun 20, 2022 3:54 PM] Number of degrees of freedom solved for: 23646 (plus 19865 internal DOFs).
[Jun 20, 2022 3:54 PM] Solution time (Study 1): 13 s.
[Jun 20, 2022 3:54 PM] Number of degrees of freedom solved for: 23646 (plus 3318 internal DOFs).
[Jun 20, 2022 3:54 PM] Solution time (Study 1): 5 s.

**Simulation numérique du champ de
température pendant le soudage
laser (Comsol Multiphysics)**

VIDEO

Conclusion générale

- Choisir le **modèle physique** à simuler.
- Contrôler la **convergence du calcul** en temps et en espace.
- Il faut faire les bonnes **hypothèses** qui correspondent aux bonnes **conditions au limites**.
- Il faut connaitre les **propriétés thermique** et les **paramètres des modèles**.