MorphoMechanX User Manual

www.MorphoMechanx.org
For further information contact:
Richard Smith, Richard.Smith@jic.ac.uk
Gabriella Mosca, gmosca@mpipz.mpg.de

1 Introduction - GUI

MorphoMechanX is an add-on of the software MorphoDynamX (www.MorphoDynamX.org) which enables to perform integrated biomechanical and morphogenetic simulations on realistic or idealised templates derived from biological samples (i.e. confocal microscopy).

In general, to run a process in MorphoDynamX and MorphoMechanX, you will need to double click on it with the mouse left button.

In Fig. 1.1 we provide a snapshot of the MorphoDynamX main window .

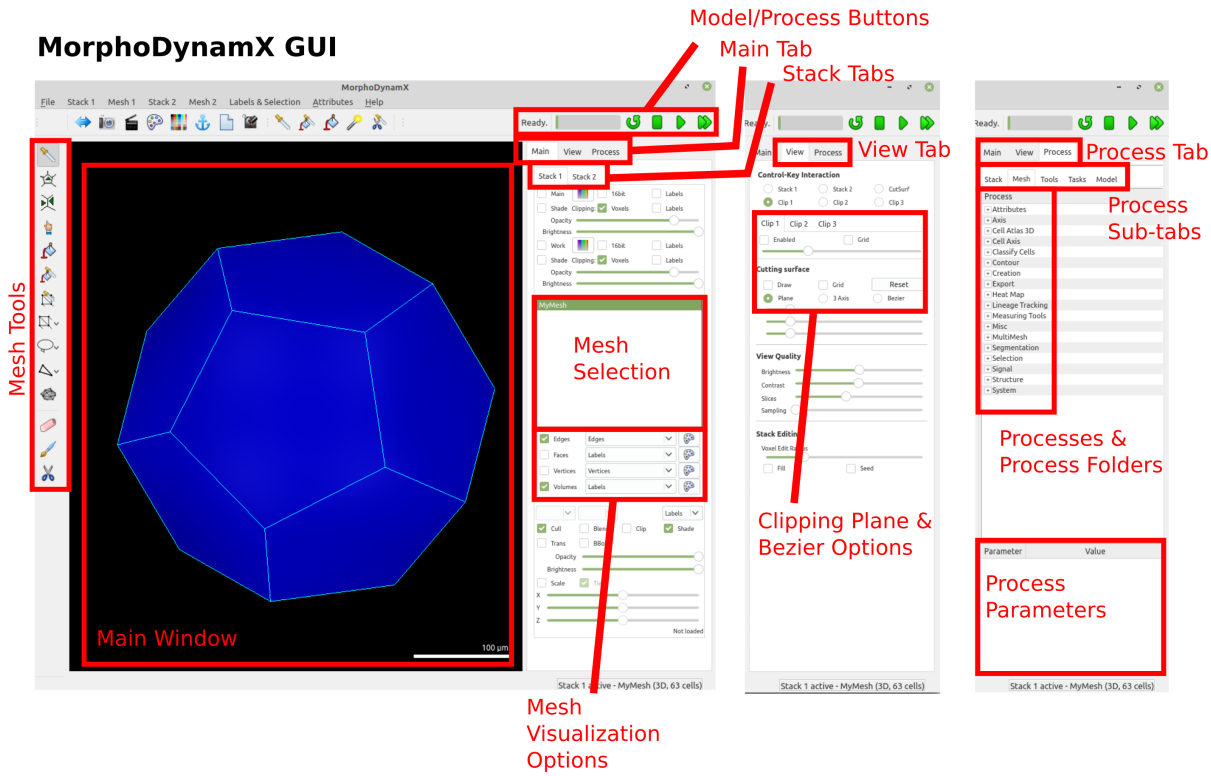


Figure 1.1: MorphoDynamX GUI with main menu, toolbar and tabs.

2 Setting and running MorphoMechanX basic models

In this section you will be guided through setting up a simple FEM mechanical simulations with membranes using MorphoMechanX. The models are presented in a progressive order of complexity for functionalities and settings and are built on top of the simpler ones. When setting up a model, some **attributes** (i.e. FEM element attribute, material attribute, pressure attribute, Dirichlet attribute) will be assigned to its mesh elements, and those create a pairing between the mesh elements (could be faces, wedges, but also vertices generally speaking) and the chosen fields (i.e. pressure, Young modulus, etc.) to which some numerical values are going to be assigned. Such attributes are, depending on the circumstances, read and written by the different model processes and then saved with the mesh (unless explicitly said otherwise in the **Attributes dialog box** on the top of MorphoDynamX window).

To be able to read properly the attributes, they need to be named consistenly (within the same name) in all the processes which generate, use or modify them.

For the models presented in the Workshop attributes, element types (it specifies whether a membrane, wedge or tetra element is being used in a certain process) and element dymensions are already assigned properly, so one can leave them as default, unless the user needs to customize the models differently. Unless stated otherwise, it is advisable to leave all the fields as default and change only the ones explicitly pointed at in the tutorial, especially if the user is not yet familiar with the framework.

Similarly as with attributes, some processes in MorphoMechanX might call other processes, so again names need to be consistent (see for example Fig. 1.2(a)), if they are not called properly, normally an error message will be displayed. The basic key options available in MorphoMechanX are the same already described in the MorphoGraphX tutorial, which we refer to. The most relevant are:

- **Alt key** to be pressed when selecting a portion of the mesh with the selection tools in the left panel of MorphoDynamX window (the Mesh Tools, see 1.1;
- **Ctrl key** to rotate only on the object selected on the Control-Key Interaction under the panel View;
- **Shift key** to translate the objects in the screen (can be combined with **Ctrl key**);
- if **Shift key** is instead used in combination with **Alt key**, it allows to sum at the newly selected objects to the pre-existing selection.

It is also important to note that in MorphoDynamX it is possible to select vertices, faces and volumes (those can be either solid elements, like wedges, or 3D cells, which are constituted by 2D faces): the selection can be done through the objects in the Mesh Tools (Fig.1.1, where the option is offereded between rectangular selection (which selects a region, whether of vertices, faces or volumes) or element selection tool (which will select directly just faces or volumes). When faces selection is active, this overrides on the visualization, the volume selection.

2.1 FEM-Inflation simulation with membranes

The model can be found under: **AutumnWorkshop/Membranes/Inflation**. To launch it, open a terminal and move into the model path. Then type: **make run**. MorphoDynamX will be launched with the model already loaded and to run it, the user will simply need to double click on **Process/Model/CCF/01 FEM Membranes**.

The model described here simulates the inflation of a multicellular template. For a reference see Sapala et al. [5]

To set up a new template, perform the following instructions.

- Launch the model as explained in the previous section.
- To load a mesh into it (or a different one than the default), go to the top menu in MorphoDynamX window and from **Mesh1/Load** select your mdxm mesh. In case you want to import a ply mesh, do **Mesh1/Import**.
- Assign the model **reference configuration** by selecting the faces of the mesh with the rectangle selection tool (see Fig. 1.2(a))and running **Model/CCF/03 Reference Configuration**, after having put in the dialog box the desired membrane thickness (if nothing is selected, the process acts on the whole mesh),

the rest can be left as default. This process will register the current mesh configuration as the reference, stress-free, configuration and will assign, in the case of membranes, their virtual thickness.

- For most model stress/strains are computed automatically in the main loop, so that generally there is no need to assign explicitly the stress/strain computation process **Model/CCF/04 Stress-Strain**, this process can be instead useful when loading a model already deformed and stresses were not saved with the mesh.

- Materials properties (for now Saint Venant-Kirchhoff material law, isotropic or transversly isotropic is available) need to be assigned. After having prescribed in the dialog box the Young moduli for the isotropic in-plane direction (Young E1E3) and fiber direction (Young E2), the Poisson ratio, run the process **Model/CCF/05 Set Material Properties**: if nothing is selected, this will apply the properties to the whole mesh, otherwise to the portion of faces selected (using the face rectangle selection tool or the pure face selection tool, see Fig. 1.2(a)).

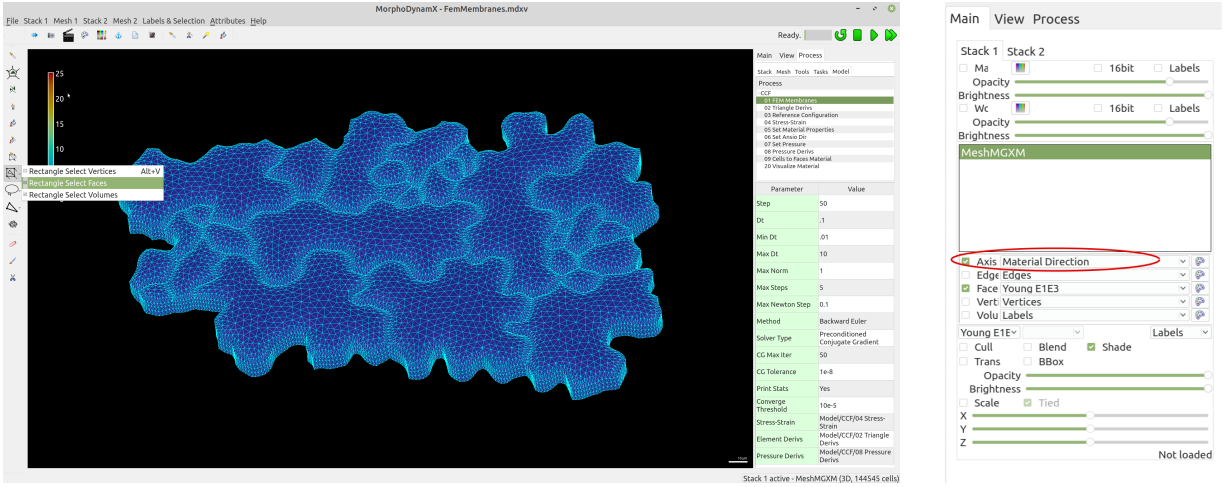
- Material properties can also be assigned on a cell-base method. To do so, select the cells with the volume selection tool (**Select Volume**) in MorphoDynamX window left-panel and select the cells for which you want to assign a specific pressure and run the same process as before (**Model/CCF/05 Set Material Properties**), but with Element dimension set to 3. Then, after assigning all the desidered cell-based material properties, run **Model/CCF/10 Cells to Faces Material**.

- If the user wants to assign anisotropic material properties, it is necessary to set an orientation. For this model it is possible to assign only global orientations (in more complex models the orientation can be based on the gradient of a morphogenetic field). To do so, select the portion of the mesh for which the orientation needs to be assigned with the usual face selection tools. In the process **Model/CCF/06 Set Aniso Dir** specify the **Direction Type**, which in this case is **E2** (the alternative being KPAR, which is for growth anisotropy), the **Direction** vector (expressed in global coordinates), whether the projection will be Parallel or Orthogonal to the specified direction (**Projection Type**) and the projection tolerance (this number sets the tolerance to decide whether the projection of the anisotropy vector on the face element is so small that the process has fails and isotropic material properties, given by the average of the two Young moduli, need to be assigned), then run the process. It is advisable to check the result of the anisotropy direction assignment by running the two sub-processes in **Model/CCF/17 Directions/**(you will need to select in the **Main** menu, in the dialog box the option **Axis** and within it, **Material Directions**, see Fig. 1.2(b)).

- To inflate the template, a pressure value needs to be assigned. Again it is necessary to select the faces for pressure assignment. For this type of model select the whole mesh faces with the usual tool and run the process **Model/CCF/07 Pressure/00 Set Pressure**, which will now assign a uniform pressure value to the whole selected faces (make sure they belong to closed surfaces or open, but with properly fixed boundaries, or it can't be a pressure load).

- At this point it is possible to run the main model process **Model/CCF/01 FEM Membranes**, with all the parameters initially left as default. The template will inflate (the simulation will terminate when the residual of the forces will be lower than the set tolerance, **Convergence Threshold**. Depending on the template, the mesh refinement, the material properties and pressure, this value will need to be adjusted (the smaller, the more reliable the result is for a given mesh resolution, the parameter is considered acceptable when, by reducing it, the resulting inflated template is not significantly different from the one with the unreduced parameter).

- To visualize the different fields, for example stress or strain, select **Faces** in the **Main** menu dialog box and the wanted field (see Fig. 1.2(b)).



(a) MorphoDynamX Model window

(b) Main menu detail

Figure 1.2: a) Example of MorphoDynamX window, open on the Model tab, with a detail on the selection tools on the left. By clicking for a prolonged time on the selection icons on the left tab, multiple options will be open as shown in the image. The empty rectangle is for vertex selection, while the one filled by a triangle for faces selection and the one filled by a wedge for wedge/tetra elements selection (all the ones enclosed by the projection onto the rectangle). Below are the lazo selection tool (same rules apply, but with a hand drawn encircling geometry) and the direct elements' selection tool. b) Detail on the Main window, where the YoungE1E3 attribute for faces has been selected.

2.2 Indentation simulation

The model allows performing indentation simulations (mimicking the action of a cellular force microscope, CFM, or of an atomic force microscope, AFM) on multicellular templates. The indentation is simulated through a needle with a spherical tip (whose radius needs to be specified by the user) which is indenting on the template in a specified point (also selected by the user). The indentation is performed parallel with the z-axis, by prescribing a displacement, subdivided in steps, of the indenter in such direction, so that the user needs to rotate its template accordingly (this simulates the mode of action of the CFM and AFM). The first model proposed can be found under the path: **AutumnWorkshop/Membranes/Indentation/indentSphere** and represents a spherical shell which undergoes indentation.

- To perform an indentation simulation, prepare the template as described in Sec.2.1.
- Run the process **Model/CCF/01 Fem Solver**, this will inflate the template (generally you need to do this before applying Dirichlet boundary conditions to sustain the mesh bottom during indentation).
- After the inflation, some Dirichlet boundary conditions will need to be prescribed, otherwise the mesh will simply displace away from the indenter. Select the points of the mesh you want to fix and after having inserted in the dialog box, at the field **Dirichlet labels**, a vector made by integers numbers different from 0 in the direction to fix the template (i.e. the vector 1 1 1 will fix the selected vertexes in x-y-z, 1 0 0 only in x, etc.), run the process **Model/CCF/09 Set Dirichlet**.
- Select, with the rectangle selection tool (the empty rectangle), a vertex (to be able to see what you have selected, in the Main menu, tick the option Edges (see Fig. 1.2(b)). The indenter center will be co-aligned in the x-y-direction with such point, displaced from it exactly by the indenter radius in the z-direction.
- With the vertex selection active, run the process **Model/CCF/11 Fem Indent**. You will first need to specify: i) Indenter Radius, ii) Indent Depth, iii) Indent Steps, iv) Indent Label (what matters is that this number is different from the one/s used for setting the Dirichlet condition), v) File Name.
- At this point the main process, **Model/CCF/01 Fem Membranes** can be run with the default parameters. The process will first check for convergence (taking into account the Dirichlet conditions assigned) then will indent in steps as specified by the user (it will not move to the next step until convergence has been reached). The indentation itself consists in moving the center of the indenter along the z-direction: if a point virtually penetrates the sphere representing the indenter tip, it will be displaced right out of it, by displacing the point along the z-coordinate (fix Dirichlet displacement), while x-y coordinates will be left free. Then equilibrium will be sought and if the reaction force on a point fixed by the indentation process is negative (which means it is not pushing against the indenter), the point will be released (so no fix Dirichlet conditions on it).
- The indentation force (given by the sum of the reaction force in z of all the points fixed by the indentatino process) versus the indentation depth will be saved in the file whose name has been specified by the user.

In the submodel **AutumnWorkshop/Membranes/Indentation/indentSepal** it is possible to indent iteratively on a series of vertices indicated in the main process list (so that the template does not need to be inflated before each indentation). It is also possible to **assign different turgor pressure to different cells**. To assign different cell pressure to different cells do the following:

- Use the volume selection tool (select the wedge icon in the left panel of MorphoDynamX window) and pick the cells for which you want to assign a certain (the same) pressure value, then run the process **Model/CCF/07 Pressure/01 Set 3D Cell Pressure**, with the desired pressure value assigned. Repeat this operation until all cells have been processed.
- Run then the process **Model/CCF/07 Pressure/02 Set Face Pressure From Volume**. This process will then transfer the volumetric pressure attribute to the cell faces (when a face belongs to two cells, the net pressure value will be assigned).

To sequentially indent on different vertices as specified by their index, add those indexes in the field "Vertex List" in **Model/CCF/00 Fem Membranes Indentation Sequence** and run the process.

The indentation model has been described in Mosca et al. [4].

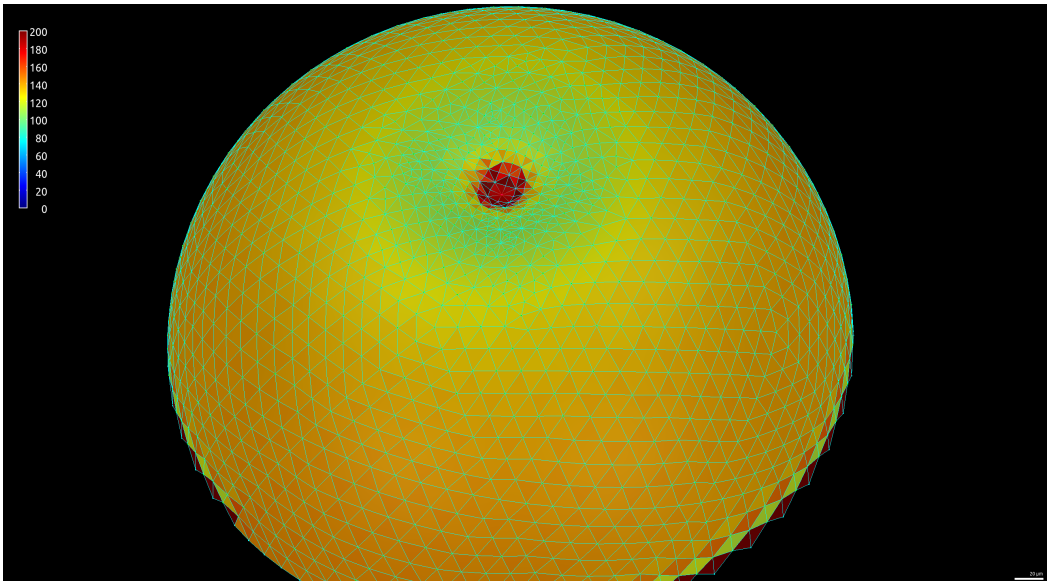


Figure 1.3: Example of an indentation simulation on a half sphere (its open edge has been fixed by Dirichlet conditions after pressurization), in red is visible the initial fixed vertex for indentation. Heatmap indicates the trace of the Cauchy stress tensor (MPa).

2.3 Extensometer simulation

In this section we see how to set up an extensometer simulation. We refer to the model **AutumnWorkshop/Membranes/Extensometer**.

- Prepare the template as described in Sec. 2.1.
- Depending on the problem modeled, you might want to set up Dirichlet conditions before or after pure pressurization has been performed. If you want to run pure pressurization before setting Dirichlet conditions, run the process **Model/CCF/01 Fem Solver**. To set Dirichlet conditions run **Model/CCF/09 Set Dirichlet** with the desired fixed labels as explained in the previous section.
- After setting Dirichlet condition, another set of Dirichlet conditions need to be assigned on the points which will be pulled by the extensometer simulation. To do so, select the points of interest and run **Model/CCF/09 Set Dirichlet**, this time all the directions need to be fixed (so all different from 0) and, importantly the label needs to be the same, and different from the one used for normal Dirichlet assignment process (i.e. if for the Dirichlet process you set 1 1 0, for the points you want to extend, assign something like 5 5 5).
- In the process **Model/CCF/09 Fem Extend** fill the dialog box making sure that the **Extensometer Label** corresponds to the one used in the previous step (so following our example, the label should be 5).
- Run the main process **Model/CCF/01 Fem Membranes**, this after convergence has been reached, will perform and extension simulation by displacing the prescribed points by the user selected amount and direction. A file will be saved storing total displacement reaction force VS displacement.

2.4 2D Growth Continuous

The model under the path **AutumnWorkshop/Membranes/growthContinuous/OvuleGrowth** allows to grow a 2D flat membrane according to user specified fields of growth and directionality. It enables the user to also assign gradient of tissue material stiffness to explore how they affect the growing structure. The model is taken from Hernandez-Lagana et al. [3] and, as built, represents the growth, in continuous tissue approximation, of a longitudinal section of an *Arabidopsis* ovule primordium. To simply run it, double-click on **Process/Model/CCF/01 FEM Membranes**. It is possible to visualize the pre-assigned fields by running

the processes **Process/Model/CCF/20 Visualize Material** and **Process/Model/CCF/21 Visualize Growth** (as usual they need then to be selected in the "Main" tab, in this case relative to the "Faces" tab). It is also possible, even with the deformed mesh, to visualize the anisotropy directions (for growth and material) by consecutively running the processes **Process/Model/CCF/17 Compute Current Directions** and **Process/Model/CCF/18 Visualize Current Directions** and setting as active the chosen field (Growth Direction or Material Direction for "Axis" in the "Main" tab, see also Fig. 1.2(b)).

To prepare a new template, proceed as follows.

- Set reference configuration, material properties (if uniform), anisotropy (if uniform) and Dirichlet as described in the previous models (Inflation and indentation models, Sec. 2.1 and 2.2).
- To partially mimic the effect of pressure, it is possible in this model to set pressure on the boundary edges of the mesh by selecting them (vertex selection tool active, rectangle in the left panel of the Main tab) and running **Process/Model/CCF/09 Set Pressure/09b Set Edge Pressure**.
- To assign growth, material and anisotropy fields based on a (pseudo-)morphogenetic diffusion, it is first necessary to set a morphogenetic diffusion process. This is contained in the submodules **Process/Model/CCF/Fem Diffusion Anisotropy** and **Process/Model/CCF/Fem Diffusion Growth**. For simplicity, we will refer to the "Fem Diffusion Growth" process in the following.
  - Select the face elements over which the diffusion process will be defined and create the diffusion element by running **Process/Model/CCF/Fem Diffusion Growth/04 Create Diffusion Element**.
  - Define the Dirichlet boundary conditions for the diffusion process (they have nothing to do with the Dirichlet boundary condition for mechanics): this will assign the nodes with fixed concentration in the simulation. Select the vertices to fix with a certain concentration value, in **Process/Model/CCF/Fem Diffusion Growth/03 Set Diffusion Dirichlet** specify the value in the field "Values" and run it.
  - Define the diffusion equation by setting values for "Diffusion Constant", "Production Constant" and "Decay Constant" in **Process/Model/CCF/Fem Diffusion Growth/01 Diffusion Derivatives**.
  - Before running, make sure that Element Type, Element Attribute, Dirichlet Attribute, Morphogen Attribute etc., are consistently set in the processes. Make also sure the Solver Type is set to "Diffusion Direct".
  - Solve the diffusion equation by running: **Process/Model/CCF/Fem Diffusion Growth/00 Diffusion Solver**.
  - Visualize the result by running **Process/Model/CCF/Fem Diffusion Growth/05 Morphogen Visualize** and make note of the name assigned to the field "Morphogen Signal" (which is also what will need to be turned in the Main model tab for the faces to see the result).
- Once the diffusion equation has been solved, it is possible to use the signal field (the name assigned to "Morphogen Signal" in the previous process) to assign a non-homogeneous growth field (it can be done similarly for material properties). For the underlying homogeneous fields, assign growth values for parallel growth (KPar), orthogonal (KPer), isotropic (Iso), strain-based (KPar-Strain, KPer-Strain, KIso-Strain) and their corresponding strain-thresholds in **Process/Model/CCF/11 Set Growth/11a Set Growth Uniform**. The assigned values will work in an additive fashion (with the exception of threshold) with the morphogenetic ones. To assign the (pseudo-)morphogenetic dependent growth, go to **Process/Model/CCF/11 Set Growth/11b Set Growth Morphogens (on Faces)** and set in the field "Morphogen Signal 1" the name assigned to "Moprhogen Signal" in the diffusion process. If multiple diffusive processes are required, it is possible to add other two (the output will be additive). Choose the scaling function for the signal and fill the other values accordingly (if only one morphogenetic signal is used, fill only the pertinent fields and leave the others empty).
- To assign material/growth anisotropy based on morphogens, one has to follow analogous steps as the ones described above for the generation of the diffusion field (it can be the same field defined for growth, or a

different one, with a different signal name and attributes, depending on the user needs). The signal name (it can be up to 3) will have to be inserted in the field "Morphogen Signal 1" (and subsequent if more fields are present) in **Process/Model/CCF/08 Set Anisotropy/08b Set Aniso Dir Morphogens**. The other fields are similar to the uniform Set Anisotropy Direction process, already described in Sec. 2.1. In this case, assigning anisotropy field based on morphogens creates a gradient of a signal which is the arithmetic average of the signals inserted, from which the direction is computed (Parallel or Orthogonal) and it overrides any pre-assigned direction computed with **Process/Model/CCF/08 Set Anisotropy/08a Set Aniso Dir Uniform**.

- Once all the fields have been assigned the simulation can be run through **Process/Model/CCF/01 FEM Membranes**, the template will be deformed by the edge pressure (if present) then grown as specified by the user, a new equilibrium configuration will be found and the process will go on iteratively.

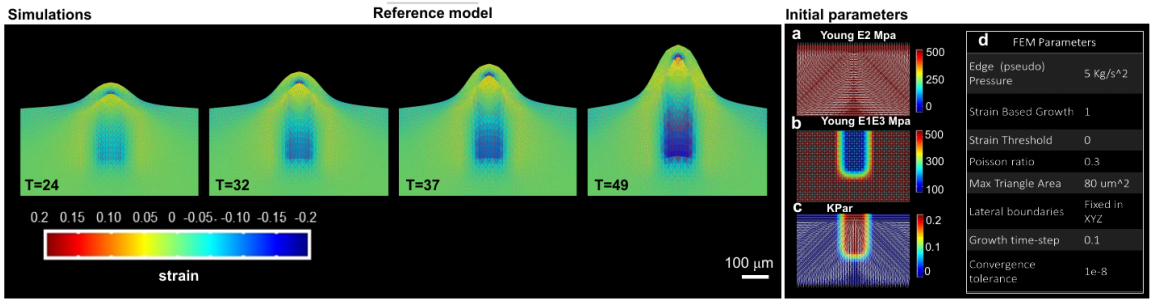


Figure 1.4: 2D continuous growth of an ovule primordium, longitudinal section (heatmap trace of Green-Lagrange strain). The initial settings are also shown on the right. Figure taken from Hernandez-Lagana et al. [3].

2.5 3D Multicellular growth simulation with morphogenetic fields

In this section we describe how to prepare a model which features growth on a multicellular template making use of polarization fields for growth amount and anisotropy. The model can be found under the path **AutumnWorkshop/Membranes/growthCells/Embryo**. The predefined model describes an *Arabidopsis* embryo undergoing a strain-based growth modulated by a diffusive field. Anisotropic material properties, aligned with the embryo circumferential direction have been assigned. By re-assignign such material properties to isotropic, the user can easily compare the effect of material anisotropy on strain-based growth directionality.

The initial template preparation can be done similarly to what explained in Sec. 2.1 and Sec. 2.2.

- The growth field based on morphogens ("KStrainIso scaling factor 1" in this case) has been assigned as described in Sec. 2.4.

- Anisotropy was assigned as a uniform field (**Process/Model/CCF/08a Set Aniso Dir**), with "Projection Type" set to "Orthogonal" w.r.t the chosen direction ("0, 0, 1").

- Since this model features 3D cells, it is also possible to assign the morphogen-diffusion Dirichlet condition cell-based, by selecting the whole cell with the volume selection tool (see the wedge in the red circle in the left panel of MorphoDynamX window in Fig. 1.5), then selecting the faces corresponding to the volume by running **Process/Mesh/Selection/Select Faces of Volumes** and then the corresponding vertices by running **Process/Mesh/Selection/Select Vertices of Faces**.

- In case one wants to assign growth parameters individually for each cell, it is possible to use the volume selection tool (the wedge symbol), select the wanted cells and run **Model/CCF/10 Growth/00 Set Growth**, with the element dimension set to 3. Make sure no face elements are selected at the same time. Then run **Process/Model/CCF/10 Growth/03 Cells to Faces Growth**. This process can be used in combination with the faces one, the resulting fields will be determined by the value chosen in the field: "Combine Method".

- By running **Process/Model/CCF/01 Fem Membranes** the template will be first inflated till equilibrium, then will growth based on the rule set by the user and a new equilibrium will be computed. This will go on iteratively, with mesh saved at the iteration times specified by the user in the main process in the field "Simulation Time To Save Mesh", with a filename specified in the field "Save Mesh Process Name".

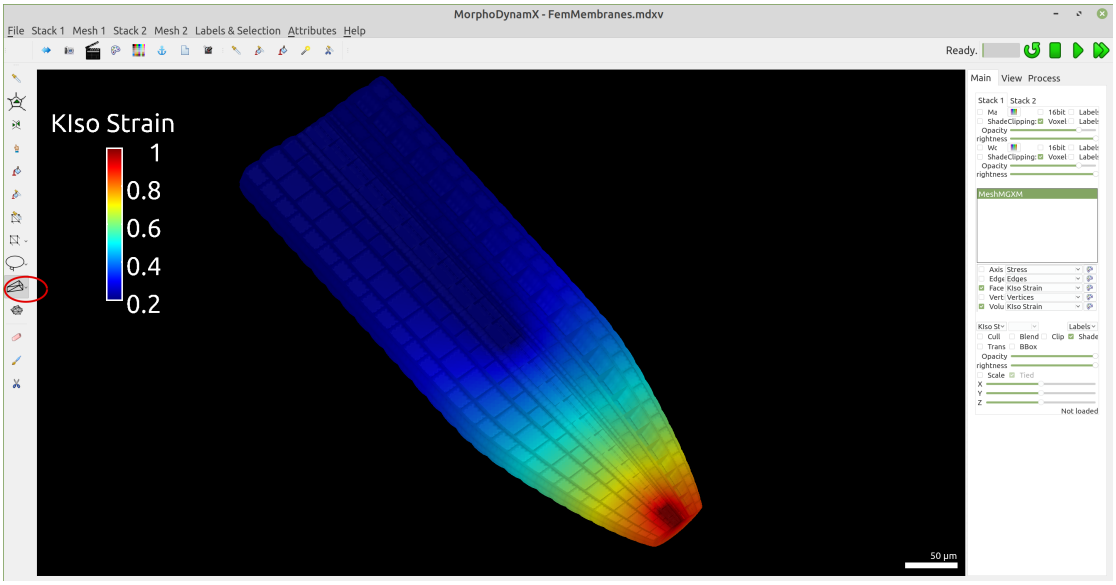


Figure 1.5: Result of a diffusive simulation on a multicellular template as shown through a longitudinal section of the template. Heatmap represent the morphogenetic modulated amount of specified isotropic strain-based growth (a-dimensional units). In the red circle, the volume selection tool, useful to select whole cells. Simulation adapted from Bassel et al. [1]

3 Continuous growth simulation with solid elements

When the user aims at modeling a whole tissue or organ, constituted by a moltitude of cells, cellularized description is normally not feasible, nor very informative (too much detail), while membranes might not be adequate (unless special symmetries are invoked). Various plant organs grow mainly in a planar (but not flat) fashion, as for example leaves, sepals and petals. For such descriptions a continuous modeling approached constitued by solid elements like wedges is a viable option.

In the following example, the margin growth of an idealised leaf is assigned, so to explore how residual stresses can break planar symmetries in growing organs. The model is under the path **AutumnWorkshop/Wedges/growthContinuous/LeafMarginPattern**.

As shown in Fig.1.6, growth is assigned parallel to the margin, with an intensity degrading from it. Instead of a proper diffusive process (which could have also been used), a distance field has been assigned (**Process/Model/CCF/Distance Field/01 Distance From Selected Vertices**. This process computes the geodetic distance of the mesh points from a set of selected vertices, with a method taken from Crane et al. [2]. It is possible to rescale the signal from a max and min value, assign a cutoff as well as assign the highest value to the minimal distance and minimal to the maximal.

Both the distance field and the diffusion field method used in wedges simulations, as in they are thought to describe planar (even if not flat) processes, are primarily defined on a surface. So the user needs to either select the vertices (for the distance field computation) or faces(for the diffusion field element creation **Process/Model/CCF/Morphogen 1/04 Create Diffusion Element**) all lying on the same surface (outer or inner) of the wedge template. The computed distane and diffusion will be computed on the surface to which either the selected vertices or the faces belong to, but the signal will be copied to volumes as well (if the option is active), so that it can be used to operate on the wedge elements.

All the other model settings can be operated analougosly to what described before, provided that the user adjust the element type to “Linear Wedge“ and element dimension to 3.

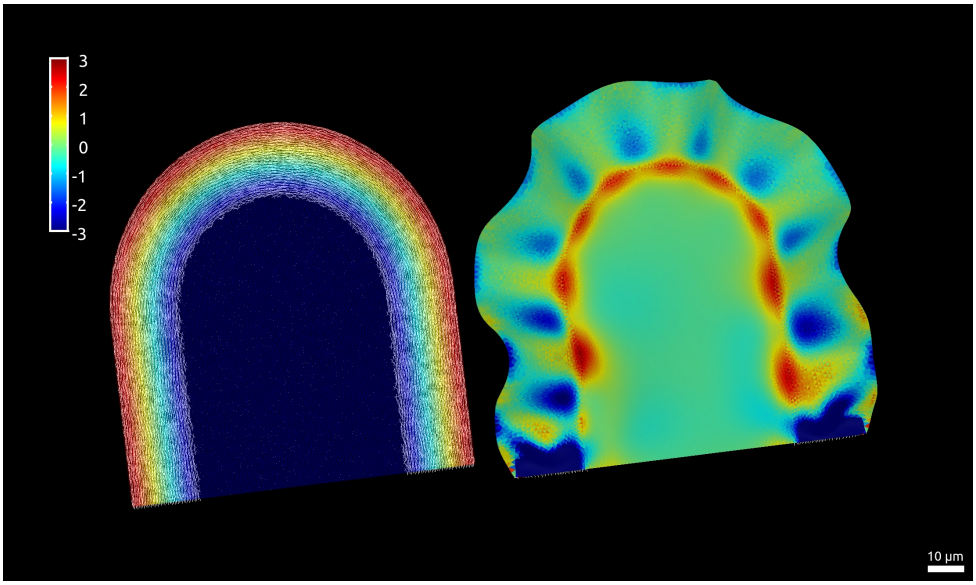


Figure 1.6: Growth simulation of a leaf margin. Left: amount and directionality (white lines) of assigned growth (KPar). Right: Grown margin of the leaf, showing the breaking of 2D symmetry of the initial template due to the accumulation of residual stresses. Heatmap: trace of Cauchy stresses, related to the right image panel.

Bibliography

[1] George W Bassel, Petra Stamm, Gabriella Mosca, Pierre Barbier de Reuille, Daniel J Gibbs, Robin Winter, Ales Janka, Michael J Holdsworth, and Richard S Smith. Mechanical constraints imposed by 3d cellular geometry and arrangement modulate growth patterns in the arabidopsis embryo. *Proceedings of the National Academy of Sciences*, 111(23):8685–8690, 2014.

[2] Keenan Crane, Clarisse Weischedel, and Max Wardetzky. Geodesics in heat: A new approach to computing distance based on heat flow. *ACM Transactions on Graphics (TOG)*, 32(5):1–11, 2013.

[3] Elvira Hernandez-Lagana, Gabriella Mosca, Ethel Mendocilla-Sato, Nuno Pires, Anja Frey, Alejandro Giraldo-Fonseca, Caroline Michaud, Ueli Grossniklaus, Olivier Hamant, Christophe Godin, et al. Organ geometry channels reproductive cell fate in the arabidopsis ovule primordium. *Elife*, 10:e66031, 2021.

[4] Gabriella Mosca, Aleksandra Sapala, Soeren Strauss, Anne-Lise Routier-Kierzkowska, and Richard S Smith. On the micro-indentation of plant cells in a tissue context. *Physical biology*, 14(1):015003, 2017.

[5] Aleksandra Sapala, Adam Runions, Anne-Lise Routier-Kierzkowska, Mainak Das Gupta, Lilan Hong, Hugo Hofhuis, Stéphane Verger, Gabriella Mosca, Chun-Biu Li, Angela Hay, et al. Why plants make puzzle cells, and how their shape emerges. *Elife*, 7:e32794, 2018.