1. Import geometry to the folder CASE/constant/triSurface
2. Create background Block Mesh – this is what is refined during the process.
	1. Copy an old blockMeshDict to CASE/system
		1. Update to give a block mesh that is as big as the area of the case, ensuring the dimensions are right on the right axes.
			1. Note that when defining the block vertices, it seems that you need to vary Z last.
	2. Copy an old controlDict to CASE/system
	3. Run command: blockMesh
	4. Run command: touch CASENAME\_block.foam
	5. There should now be a folder CASE/constant/polyMesh with files in there
	6. Inspect in ParaView, along with the geometry of the case, to ensure they align
3. surfaceFeatureExtract

This deals with the sharp edges of the geometry

* 1. Copy an old surfaceFeatureExtractDict to CASE/system
	2. Update so that there is one entry per stl file in the body of the file
	3. Run command: surfaceFeatureExtract
	4. There should now be emesh files in CASE/constant/triSurface folder
1. Decompose
	1. Needed if running meshing in parallel, which I didn’t do. See video from 24:00.
2. snappyHexMeshDict
	1. Copy an old snappyHexMeshDict to CASE/system
	2. Copy old versions of fvScheme, fvSolution, meshQualityDict to CASE/system. These probably don’t need to be changed.
	3. castellatedMesh = true
	4. snap = true
	5. addLayers = false to begin with but can make this true once things look good
	6. geometry: add all geometry files, ensuring the name reflects the patch name you want. Also add any areas for refinement (I’ve called it refinementBox).
	7. castellatedMeshControls:
		1. I’ve just taken the default values for the first five lines here (100000, 2000000, 10, 0.10, 3).
		2. features:
			1. This does refinement around the eMesh files created at the surfaceFeatureExtract step. I’ve turned this off because it gives refinement around the inlets and roof, which I don’t want. (Previously, had: level: setting inlet, outlet and sides = 0 means no refinement where these surfaces join. You still get refinement where these surfaces join ground and clamp given they are = 6.)
		3. refinementSurfaces:
			1. include all patches that you want to limit the refinement on. This can be a limit of maximum or minimum.
			2. level: setting to 0 0 means no refinement, 5 5 means high refinement, 0 5 means can vary
		4. resolveFeatureAngle
			1. default is 30. I’ve set to 100 to stop the roof from refining in its corner.
		5. refinementRegions:
			1. add boxes for the areas you want to refine in more details. They need to listed in the geometry section
			2. mode = inside means refine within that area
			3. levels ((1E15 3)): no idea what the 1E15 means, but the 3 changes the size of the smallest refinement.
	8. Make an empty folder CASE/0
		1. If there is already files in a 0 folder (eg, p and U), rename this first (eg to 0\_orig)
	9. Run command: snappyHexMesh -overwrite
3. Once meshing complete:
	1. touch CASE\_mesh.foam
	2. inspect in Parafoam
	3. From the folders that appeared during mesh, open the one with the highest number, copy the folder polyMesh, then save that to CASE/constant.
	4. Delete these numbered folders
	5. In the CASE/constant/polyMesh folder, check that the boundary file only contains the boundaries you want – that none have come over from the original blockMesh.